Automation

5600 CNC Mill

Instructor Guide
05618-10

Printed in Canada

FIRST EDITION

First Printing, November 1999

ISBN: 0-86657-148-5

Printed in the United States of America

Lab-Volt License Agreement

By using the software in this package, you are agreeing to become bound by the terms of this License Agreement, Limited Warranty, and Disclaimer.

This License Agreement constitutes the complete agreement between you and Lab-Volt. If you do not agree to the terms of this agreement, do not use the software. Promptly return the compact disk and all other materials that are part of this product within ten days to Lab-Volt for a full refund or credit.

1. License Grant. In consideration of payment of the license fee, which is part of the price you paid for this, Lab-Volt, as Licensor, grants to you, the Licensee, a nonexclusive, nontransferable license to use this copy of the Mill Level 4 Windows® 95/NT software. Lab-Volt reserves all rights not expressly granted to the Licensee.

2. Ownership. As the Licensee, you own the physical media on which the Mill Level 4 Windows 95/NT software is originally or subsequently recorded or fixed, but Lab-Volt retains title to and ownership of the software programs recorded on the original disk and any subsequent copies of the Mill Level 4 Windows 95/NT software, regardless of the form or media in or on which the original and other copies may exist. This license is not a sale of the original software program of the Mill Level 4 Windows 95/NT software or any portion or copy of it.

3. Copy Restrictions. The Mill Level 4 Windows 95/NT software and the accompanying materials are copyrighted and contain proprietary information and trade secrets of Lab-Volt. Unauthorized copying of this Mill Level 4 Windows 95/NT software even if modified, merged, or included with other software or with written materials is expressly forbidden. You may be held legally responsible for any infringement of Lab-Volt's intellectual property rights that is caused or encouraged by your failure to abide by the terms of this agreement. You may make copies of the Mill Level 4 Windows 95/NT software solely for backup purposes provided the copyright notice is reproduced in its entirety on the backup copy.

4. Permitted Uses. This Mill Level 4 Windows 95/NT software, 5600 CNC Mill Level 4 Instructor Guide, and all accompanying documentation is licensed to you, the Licensee, and may not be transferred to any third party for any length of time without the prior written consent of Lab-Volt. You may not modify, adapt, translate, reverse engineer, decompile, disassemble, or create derivative works based on the Mill Level 4 Windows 95/NT software without the prior written permission of Lab-Volt. Written materials provided to you may not be modified, adapted, translated, or used to create derivative works without the prior written consent of Lab-Volt.

5. Termination. This agreement is effective until terminated. It will terminate automatically without notice from Lab-Volt if you fail to comply with any provisions contained herein. Upon termination you shall destroy the written materials, the Mill Level 4 Windows 95/NT software, and all copies of it, in part or in whole, including modified copies, if any.

6. Registration. Lab-Volt may from time to time update the Mill Level 4 Windows 95/NT software. Updates can be made available to you only if a properly signed registration card is filed with Lab-Volt or an authorized registration card recipient.

7. Miscellaneous. This agreement is governed by the laws of the State of New Jersey.

Limited Warranty and Disclaimer

This Mill Level 4 Windows 95/NT software has been designed to assure correct operation when used in the manner and within the limits described in this 5600 CNC Mill Level 4 Instructor Guide. As a highly advanced software product, it is quite complex; thus, it is possible that if it is used in hardware configurations with characteristics other than those specified in this 5600 CNC Mill Level 4 Instructor Guide or in environments with non-specified, unusual, or extensive other software products, problems may be encountered by a user. In such cases, Lab-Volt will make reasonable efforts to assist the user to properly operate the Mill Level 4 Windows 95/NT software but without guaranteeing its proper performance in any hardware or software environment other than as described in the 5600 CNC Mill Level 4 Instructor Guide.

This Mill Level 4 Windows 95/NT software is warranted to conform to the descriptions of its functions and performance as outlined in this 5600 CNC Mill Level 4 Instructor Guide. Upon proper notification and within a period of one year from the date of installation and/or customer acceptance, Lab-Volt, at its sole and exclusive option, will remedy any nonconformity or replace any defective disk free of charge. Any substantial revisions of this product, made for purposes of correcting software deficiencies within the warranty period, will be made available, also on a licensed basis, to registered owners free of charge. Warranty support for this product is limited, in all cases, to software errors. Errors caused by hardware malfunctions or the use of non-specified hardware or other software are not covered.

LICENSOR MAKES NO OTHER WARRANTIES OF ANY KIND CONCERNING THIS PRODUCT, INCLUDING WARRANTIES OF MERCHANTABILITY OR OF FITNESS FOR A PARTICULAR PURPOSE. LICENSOR DISCLAIMS ALL OBLIGATIONS AND LIABILITIES ON THE PART OF LICENSOR FOR DAMAGES, INCLUDING BUT NOT LIMITED TO SPECIAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF OR IN CONNECTION WITH THE USE OF THE SOFTWARE PRODUCT LICENSED UNDER THIS AGREEMENT.

Questions concerning this agreement and warranty and all requests for product repairs should be directed to the Lab-Volt field representative in your area.

LAB-VOLT SYSTEMS, INC.
P.O. Box 686
Farmingdale, NJ 07727
Attention: Program Development
Phone: (908) 938-2000 or (800) LAB-VOLT
Fax: (908) 774-8573
Technical Support: (800) 522-4436
Technical Support E-Mail: techsupport@labvolt.com

Table of Contents

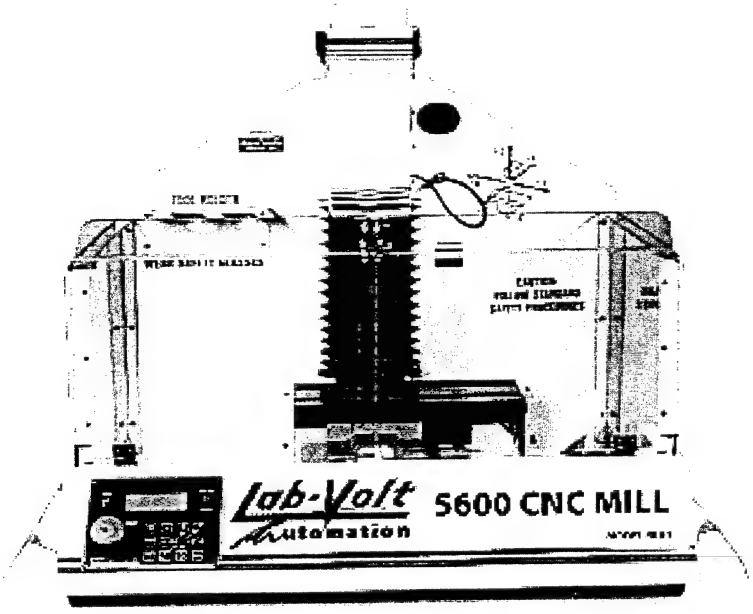
Table of Contents

Table of Contents

Table of Contents

INTRODUCTION

The Lab-Volt Automation 5600 CNC Mill is a computer controlled milling system designed as an educational tool for training students in computer aided design (CAD), computer assisted manufacturing (CAM), and computer numerical controlled (CNC) milling. It can be used to mill small parts to programmed specifications in a variety of materials and can also be connected to robotic devices to encompass a number of computerized manufacturing systems including computer integrated manufacturing (CIM) and flexible manufacturing systems (FMS).



5600 CNC Mill

This document is a description of the procedures needed to set up the 5600 CNC Mill with the Level 4 software and a brief description of the curriculum. It covers the following subjects:

- Mill specifications
- Mill setup
- Computer requirements
- Level 4 software installation
- Safety considerations
- Activity objectives, tests, and answers

INSTALLING THE 5600 CNC MILL

Check the Parts

After you open the shipping container containing the Lab-Volt Automation 5600 CNC Mill, you should find the Packing List (TLA 93643-10) in an envelope on top of the mill. The contents of the packing list are given in the table below.

5600 CNC Mill Packing List

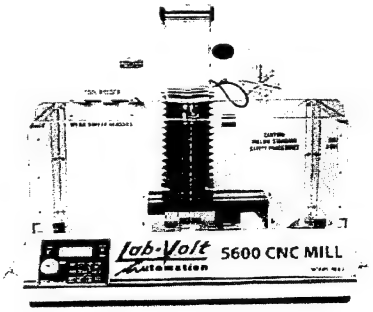
Indent. No.	Description
AS 95804-10	MILL COMPLETE DM 5600-10
SW 5647-00	CNC MILL LEVEL 4 Software
DM 5404-00	WAX PACKAGE
TA 14126-00	WARRANTY CARD
TL 174-00	NOTICE LOSS/DAMAGE

You receive only one of each part in the list, unless more than one is shown. Please check to be sure that you have received all the parts. If there is a shortage, please contact our Technical Support Department at 1-800-522-4436, and we will be happy to assist you.

Mill Complete DM 5600-10 (AS 95804-10)

> **CAUTION:** Removing the headstock support can cause headstock damage that is not covered by the warranty. Do not remove the headstock support until instructed to do so.

The Lab-Volt Automation 5600 CNC Mill is enclosed in a cabinet made of Lexan® plastic. The mill is delivered with a 1/8" end mill and a R8 collet with setscrew factory installed.



5600 CNC Mill

The following pages list the items that are shipped with the 5600 CNC Mill (95801-10). The items are broken up into three categories: Level 4 Mill Software, Tool Package, and Mill Accessories. The items in each category are shipped together.

CNC Mill Level 4 Software (SW 5447-00)

Indent. No.	Part Name	Description
AV 93903-00	CD CNC Level 4	This is the CD-ROM for installing the Level 4 Software for Mill and Lathe Version 4.0.
TM 05618-00	5600 CNC Mill Student Manual	The 5600 CNC Mill Student Manual contains a description of the CNC milling operations and nine student activities.
TM 05618-10	5600 CNC Mill Instructor's Guide	The 5600 CNC Mill Instructor's Guide that contains instructions for installing the mill, a summary of the student activities, tests, and answer keys.
AX 5390-04	Mill Hardlock Parallel Port	A personal computer parallel port hardlock key connector that the PC requires for the Mill Level 4 software to communicate with the 5600 CNC Mill.
TA 14126-00	Data Sheet Warranty	Warranty Card.
SE 93601-00	Binder 3 Ring	A binder to house the CNC Mill documentation.
TI 93605-00	Notice Firmware Update	Contains important information regarding the CNC 5600 firmware.

Tool Package

This is a list of the tools that accompany the 5600 CNC Mill.

Indent. No.	Part Name	Description
JM 96806-00	Tool Holder 1 Set	3/8" Setscrew End Mill Holder and 3" Tension Bar.
JM 96902-00	3/16" End Mill 1 Pc.	3/16" Setscrew End Mill Holder and 3" Tension Bar.
JM 95908-00	End Mill 1/8" x 3/8"	1/8" End Mill. Used with 3/8" Setscrew End Mill Holder.
JM 96787-00	Edge Finder	Edge Finder with a 3/8" shaft.
JM 96308-00	End Mill ¼"	Used with 3/8" Setscrew End Mill Holder.
JM 95897-00	3/16" End Mill	Used with 3/8" Setscrew End Mill Holder.
JM 97024-00	Vise Piston-Grip	Mill's vise to be mounted to the mill table with the hold down set.
JM 95909-00	R8 End Mill Holder	Used with the R8 collet.
JM 95903-00	Step Block/2	Used with the serrated clamp as the hold down set.
JM 95902-00	Serrated Clamp	Used with the step block as the hold down set.
JM 95453-00	Tool Bag CNC	Used to hold the accessory tools.

Wax Package (DM 5404-00)

The wax package consists of 100 blocks of 2" x 2" x 0.5" industrial machinable wax that you use as milling stock.

Mill Accessories

Indent. No.	Part Name	Description
JM 95286-00	Chip Brush 1 inch	A 1-inch brush used to clean sock chips from mill cabinet.
JM 95339-00	Hex Key 3/16"	Used to make various adjustments on the mill.
JM 95101-00	2.5mm Allen Wrench	Used with 3/8" Setscrew End Mill Holder.
JM 9380-00	Wrench Box 625 x 750	Used with 3/8" Setscrew End Mill Holder.
JM 91825-00	Wrench Allen/32sht	Used with 3/8" Setscrew End Mill Holder.

Contents of 3 X 5 Zip Lock® Bag

Indent. No.	Part Name	Description
ML 95823-00	Fuse 3A Slow Blow - 2 pieces	Three spare fuses.
ML 95705-00	Fuse 10A Slow Blow	Two spare fuses.
CG 91363-00	ADPTR CNVTR 9M25F	9 pin to 25 pin serial cable adapter.

Required Parts for Assembly and Operation

The following table lists the parts that you need for assembling the 5600 CNC Mill and for performing the nine activities in the Student Manual.

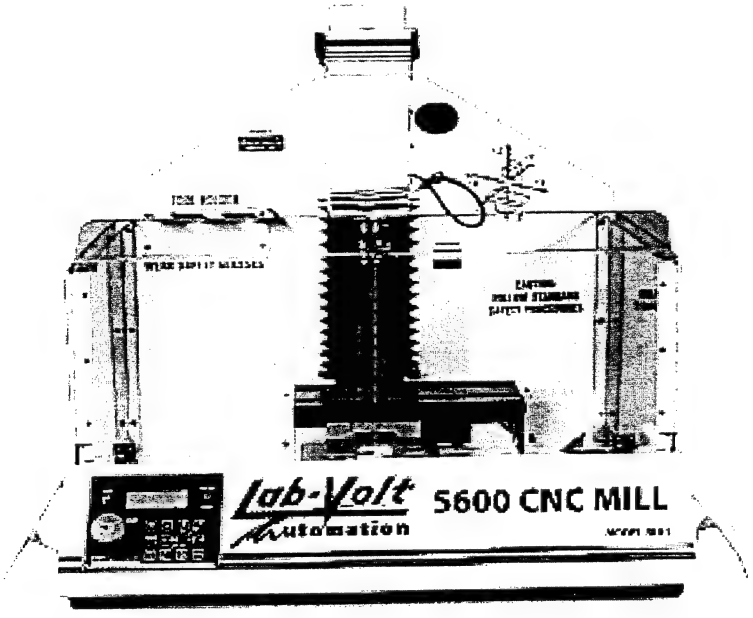
Parts Required for Assembling the 5600 CNC Mill and the Student Guide Activities

Ident. No.	Part Name	Comment
KJ 95451-00	Power Cord	For 120V power to the mill. Customer installs.
KJ 93017-00	CA DB9F-DB9F NULL MODEM	Serial cable between Mill and PC. Customer installs.
JM 95908-00	End Mill 1/8"	Installed on Mill before shipping.
JM 95909-00	End Mill Holder R8	Installed on Mill before shipping.
JM 97024-00	Vise Piston-Grip	Mill's vise to be mounted to the mill table with the hold down set.
SW 5447-00	CNC Mill Level 4 Software	Mill software for the PC. Customer Installs.

(Table continued)

Ident. No.	Part Name	Comment	Included In
AX 5390-04	Mill Hardlock Parallel Port	Hardlock key connected to the parallel port of the PC. Customer Installs.	SW 5447-00
TM 05618-00	Student Guide	Contains student activities.	SW 5447-00
TM 05618-10	Instructor Guide	Contains mill installation instructions.	SW 5447-00
DM 5404-00	Wax Package: 100 pieces 2" X 2" X .5"	Used as milling stock.	DM 5600-20
JM 95286-00	Chip Brush 1"	Used to clean stock chips from the mill cabinet.	AS 93204-00

Description of the 5600 CNC Mill



Lab-Volt 5600 CNC Mill

The 5600 CNC Mill is a fully functional mill that can be programmed from a personal computer. The Lab-Volt Automation 5600 CNC Mill is enclosed in a cabinet made of Lexan® plastic.

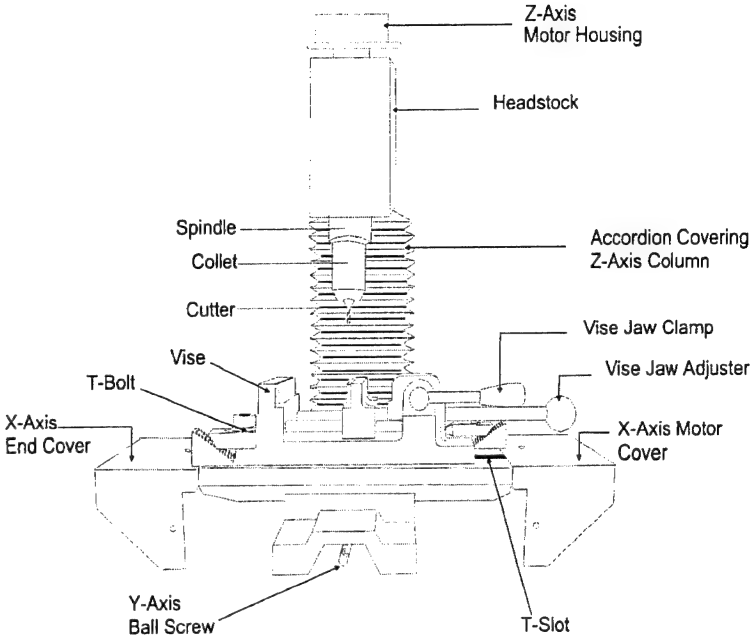
A serial cable connects a personal computer to the connection panel on the left side of the mill's cabinet. You will install the Lab-Volt *Mill Level 4* software on the computer. The *Level 4 software* is a comprehensive "design to production" application program. You can use this software to write a part program for the design of a new machine part, download the part program to the mill, and then automatically machine the part.

Level 4 Mill software can accept part programs generated by other Industrial Standard Operation (ISO) software like *Virtual Gibbs®* and *CAD\CAM Link*. The Lab-Volt 5600 CNC Mill and the *Level 4 Mill* software are powerful tools in a modern CNC training environment.

The 5600 CNC Mill contains firmware that enables the mill to accept part programs from the computer. The firmware also permits you to manually control the mill operation from the control panel screen and keypad on the right-front side of the cabinet.

The 5600 CNC Mill uses stepper motors to move the milling table along the X-axis (left and right) and the Y-axis (backward and forward). An additional stepper motor moves the headstock along the Z-axis (up and down) and a 3/4-hp dc motor drives the spindle.

A vise holds the stock on the milling table. The stock is fed against a cutting device called an end mill. The end mill is a circular tool with a series of cutting edges on its circumference. You can change the end mill to accommodate different stock materials and milling designs.



Mill Table and headstock

5600 CNC Mill Specifications

Machine Dimensions

Width	38.5 in.	97.8 cm
Depth	32.0 in.	81.3 cm
Height	29.5 in.	74.9 cm
Weight	480 lb	218 kg

Feed Motors

Stepper Motors	0.9 degrees, 400 step/rev
Rapid Traverse Speed	20 in./min (51 cm/min)

Mechanical Working Range

X (longitude)	7.5 in.	19.1 cm
Y (transverse)	4.0 in.	10.2 cm
Z (vertical)	≤8.0 in. (varies with collet)	20.3 cm

Milling Table

Table Surface	5.5 in. x 12 in.	14 x 30 cm
Hold Down T slots	2	
Slot width/spacing	0.625 in./2.5 in.	1.6/6.4 cm
Maximum clearance	8.5 in.	21.6 cm

Cutting Spindle

Throat Clearance	7.5 in.	19.1 cm
Spindle Taper	R8	

Main Spindle Drive

Speed Range (continuous)	0 – 3400 rpm
Motor	3/4 hp dc

Recommended materials for milling are wax, plastic, proto-foam, and soft metals such as aluminum and brass.

Requirements

Space

The 5600 CNC Mill requires a space that will accommodate a width of 39 inches and a height of 30 inches. It also requires a depth of 34 inches plus several inches for access to the power and communications connections. The table holding the mill must support a minimum of 500 pounds.

Electrical

120 Volts AC, 60 Hz (220V/50 Hz model available)
20 Amp service

SAFETY PROCEDURES

Introduction

This page has important safety information divided into three areas: general safety, machine safety, and safety tips. Read all safety information before beginning the activities in this manual.

General Safety

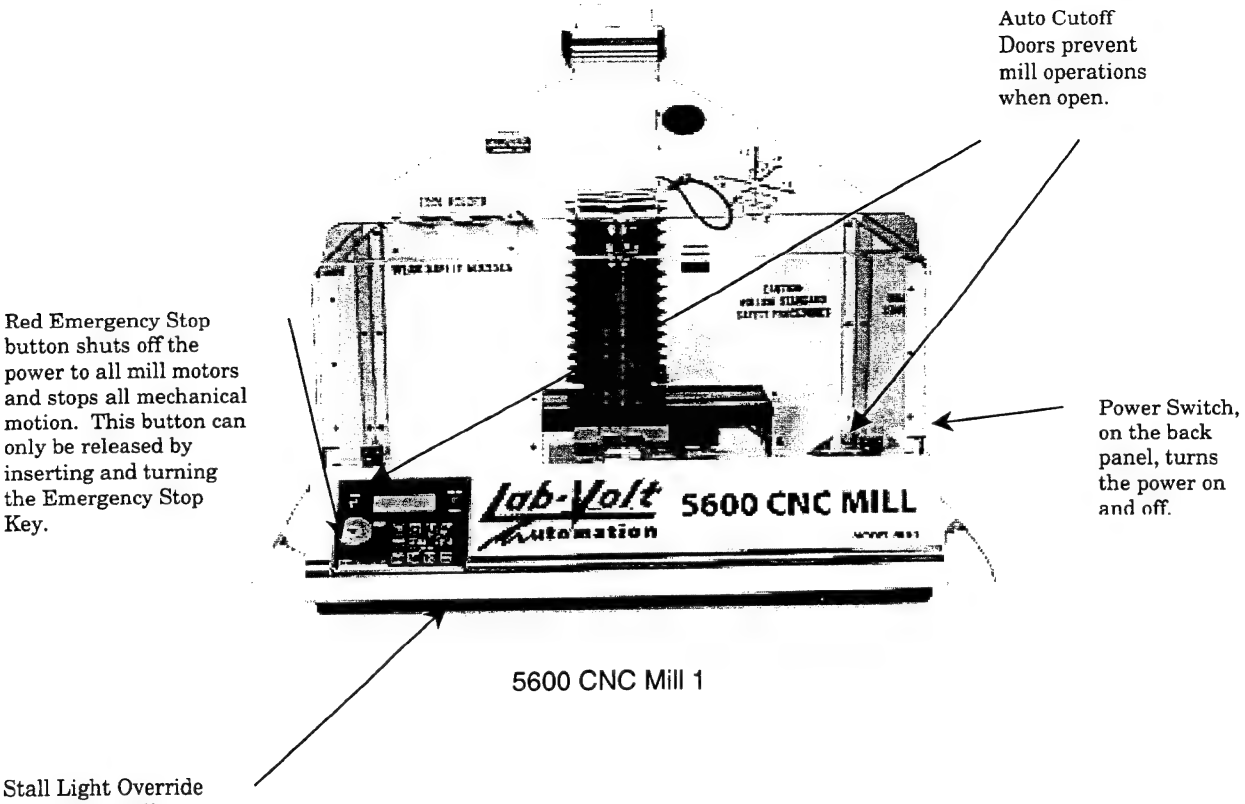
- The student must follow the safety rules for the location and shop in which they are working.
- Know the location of the shop first aid kit, what it contains, and how to use it.
- Encourage the students to pay attention to what they are doing at all times.
- Use common sense.

CNC Machine Safety

The Lab-Volt 5600 is a high-powered machine tool. It is built to machine plastic and soft metal, so there is a great deal of power behind the motors and end mill. Pay serious attention to the safety precautions listed below.

- Know what to do in an emergency BEFORE operating the mill.
- Always wear eye protection when operating the mill.
- Always close the mill door before trying to execute a program.
- Never open the mill door while the mill is running.
- After a program finishes running, wait until all mill movement has stopped before opening the mill door.
- Always keep the mill cabinet clear of nonessential materials while a part is being milled.
- Don't let the end mill run into the vise. In addition to damaging the machine, shattering pieces can also harm individuals.
- Never misuse or remove any part of the mill or mill tools.
- Never leave the mill unattended while a program is running.
- Know the emergency stop options for the mill.

The figure below shows the emergency stop options for the 5600 CNC Mill.



5600 CNC Mill 1

- Use any of the above methods to shut the mill down if a dangerous situation develops.
- Never leave the key in the *Emergency Stop* button.
- Always turn the mill off before cleaning or servicing it.

Safety Tips

- Remember that once you download the program, control is turned over to the part program. This is different from conventional machining where the operator manually controls the end mill. In CNC machining, once the mill is under its own control, the operator no longer has a choice where the tool moves.
- If a dangerous move is entered in the part program, the mill will execute that move. The mill has no ability to reason. It does not know the difference between a "good" and a "bad" move.
- Emulate all programs in multiple views before machining to check the clearance and safety of every move.

Safety Test and Answers

The following Safety Test appears in the 5600 CNC Mill Student Manual. In this Instructor Guide, the correct answers are shown in bold print.

1. The large red button located on the mill control panel is the manual speed override.

   a. True
   b. False

2. One way to halt the execution of a part program is to press the *Stall Light Override* button.

   a. True
   b. False

3. Always wear safety goggles while operating the mill.

   a. True
   b. False

4. The safety door can remain open while operating the mill if the operator wears safety goggles.

   a. True
   b. False

5. Never leave the key in the *Emergency Stop* button.

   a. True
   b. False

6. Never leave the mill unattended while it is running.

   a. True
   b. False

7. Always turn the mill off before cleaning or servicing.

   a. True
   b. False

8. If pushed, the *Emergency Stop* button can be released simply by pulling it.

   a. True
   b. False

9. Never leave any tools inside the mill cabinet.

   a. True
   b. False

10. Know what to do in an emergency situation before operating the mill.

    a. True
    b. False

Safety Test

1. The large red button located on the mill control panel is the manual speed override.

   a. True
   b. False

2. One way to halt the execution of a part program is to press the *Stall Light Override* button.

   a. True
   b. False

3. Always wear safety goggles while operating the mill.

   a. True
   b. False

4. The safety door can remain open while operating the mill if the operator wears safety goggles.

   a. True
   b. False

5. Never leave the key in the *Emergency Stop* button.

   a. True
   b. False

6. Never leave the mill unattended while it is running.

   a. True
   b. False

7. Always turn the mill off before cleaning or servicing.

   a. True
   b. False

8. If pushed, the *Emergency Stop* button can be released simply by pulling it.

   a. True
   b. False

9. Never leave any tools inside the mill cabinet.

   a. True
   b. False

10. Know what to do in an emergency situation before operating the mill.

    a. True
    b. False

ASSEMBLING THE 5600 CNC MILL

Requirements

Space

The 5600 CNC Mill requires a space that will accommodate a width of 39 inches and a height of 30 inches. It also requires a depth of 34 inches plus several inches for access to the power and communications connections. The table holding the mill must support a minimum of 500 pounds.

Electrical

120 Volts AC, 60 Hz (220V/50 Hz model available)
20 Amp service

Mill Unpacking

1. Remove the packing slip attached to the shipping container.
2. Cut the straps connecting the container to the skid and open the container.
3. Remove all 5600 CNC Mill parts.
4. Examine the packing list to ensure that all parts are included. If there is a shortage, contact the Lab-Volt Systems representative.

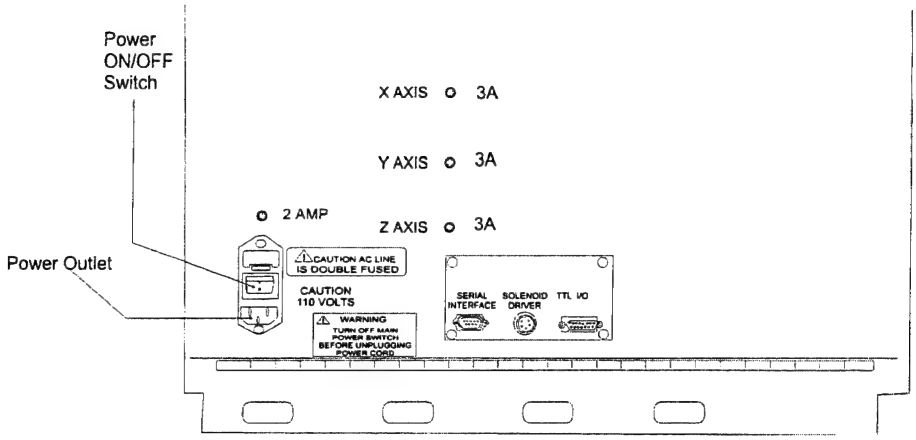
> **WARNING:** The mill weighs almost 500 pounds. Use appropriate means to transfer it from the shipping crate to its new location. Improper lifting can cause serious injury.

5. Place the mill on the table with the back panel exposed so that connections can be made.
6. Remove the packing material from the mill. Do *not* remove the headstock support yet.

> **CAUTION:** Removing the headstock support can cause headstock damage that is not covered by the warranty. Do not remove the headstock support until instructed to do so.

Mill Connections and Assembly

Connections



Back Panel

Back Panel

Power ON/OFF Switch/Outlet – For connection to an electrical power source.

SERIAL INTERFACE port – A 9-pin port for connection to a computer.

TTL I/O port – A 15-pin port for connection to a Model 5200 Robot or for providing up to four 5-volt digital input and four 5-volt digital output lines.

SOLENOID DRIVER port – A round 5-pin port for solenoid driver connections for up to four auxiliary devices; e.g., a pneumatic vise.

1. Ensure that the power switch on the back panel is in the OFF position. The symbol "O" is depressed when the power switch is in the OFF position.

   CAUTION: If the power ON/OFF switch is set to ON when the power is applied, damage can result that is not covered by the warranty. Ensure that the power ON/OFF switch is set to OFF.

2. Remove the power cable from the Accessory Pack and insert it into the power outlet under the ON/OFF switch on the back panel of the mill. Do not connect the power cable to the power source yet.

3. Remove the serial cable from the Accessory Pack and connect it to the 9-pin Serial Interface port on the back panel of the mill.
4. Connect the other end of the serial cable to the 9-pin communications port of the computer. An adapter has been provided for a computer with a 25-pin communications port. If the connected computer has more than one communications port, note the number of the port used (COM1 or COM2, etc.). The port designation is required when setting up the Level 4 software.

5. Connect the software key to the parallel (printer) port on the computer. If a printer is connected, disconnect the printer cable first and connect it to the exposed end of the software key.

   NOTE: Without the software key, the software will only operate in demonstration mode.

6. Connect the power cable to the power source (electrical outlet).

7. Remove the headstock support.

Removing and Installing the Mill Vise

The milling vise comes installed on the mill. This procedure is included if it must be removed – for milling larger pieces of stock, for example. Two T-bolts come with the milling vise. Each T-bolt consists of a nut, screw, washer, bracket, and T-nut.

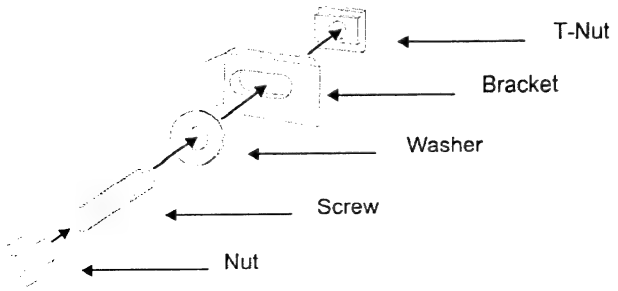
To assemble the T-bolt:

1. Put the nut on one end of the screw.

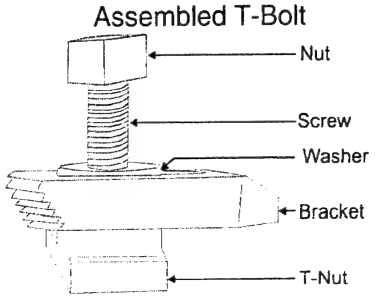
2. Slide the washer then the bracket over the other end.

3. Screw the T-nut onto the screw.

T-Bolt Components



Assembled T-Bolt



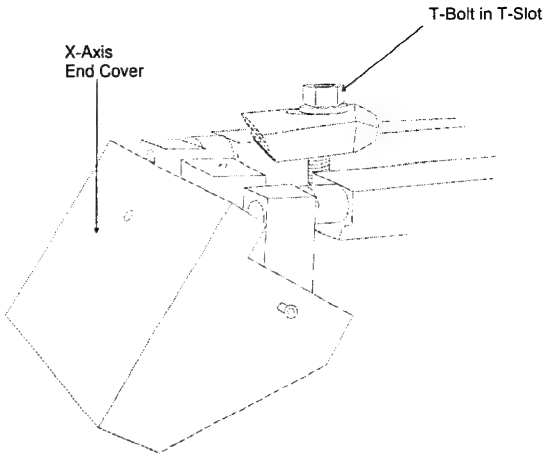
T-Bolt Components

Assembled T-Bolt

To install the T-bolt into the T-slot:

1. Using the 2.5 mm hex wrench, unscrew the top screw securing the X-axis end cover on the left side of the mill.

2. Loosen the X-axis end cover side screws just enough so that the cover can be tilted back and away from the mill table.



X-Axis End Cover Flipped Back

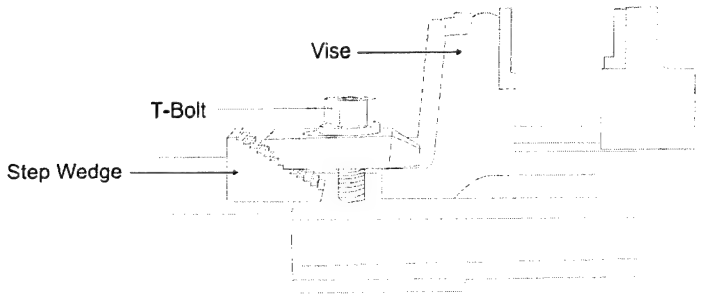
X-Axis End Cover Flipped Back

3. With the T-nut below and the bracket above the milling table, slide one T-bolt into each of the T-slots.

4. Place the vise on the milling table between the T-bolts. Align the vise so that it is centered over the T-slots and parallel with the front edge of the table.

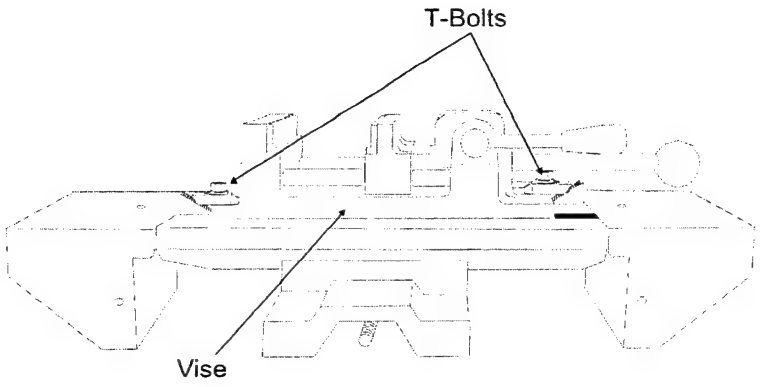
5. Slide the T-bolts to either side of the vise so that the brackets are over the vise edges.

6. Push the step wedge up to the other side of the T-bolt so that the bracket is held parallel to the mill table.



Step Wedge and T-Bolt Securing Vise

7. Tighten the bolts to secure the vise to the milling table.



Vise and Mill Table

MILL LEVEL 4 SOFTWARE

Requirements

An Intel-based or comparable personal computer with the following minimum features:

- 486DX-66 CPU (Pentium® class recommended)
- Windows® 95, Windows 98, or Windows NT 4.0
- 16 MB of RAM
- Hard drive with 15 MB of free space
- 4X CD-ROM drive or higher
- VGA display with 256 or more colors at 640 x 480 (800 x 600 recommended)
- RS-232 serial port for communications with mill hardware
- Parallel port for the Hardlock Key and for printing/plotting

Software Installation

1. Place the Level 4 Software for Mill and Lathe Version 4.0 CD-ROM in the CD-ROM drive.

   For some personal computer systems, just inserting the Level 4 Software for Mill and Lathe Version 4.0 CD-ROM in the CD drive brings up the Level 4 CNC Setup screen.

   If the Setup screen appears after inserting the CD-ROM, click on *Install Level 4 Software*. After a few seconds, the Level 4 CNC Welcome screen will appear. Go to the following step 8 and proceed with the software installation.

   If the Setup screen does not appear after inserting the CD-ROM, proceed with step 2 below.

2. On the Windows 95, 98, or NT 4.0 Desktop, double click the **My Computer** icon. (The My Computer window appears.)

3. Double click the **Control Panel** icon.

   The Control Panel window appears.

4. Double click the **Add/Remove Programs** icon.

   The Add/Remove Program Properties screen appears.

5. Click the **Install/Uninstall** tab, and then click the **Install** button.

   The Install Program From Floppy Disk or CD-ROM screen appears.

6. Click the **Next** button.

   The Run Installation Program screen appears with the command line E:\SETUP.EXE for the installation program where E:\ is the CD-ROM drive.

7. Click the **Finish** button.

   After a few seconds, the Level 4 CNC Welcome screen appears. Read the Welcome screen. It is strongly recommended that you exit all Windows programs before running the Setup program.

8. Click the **Next** button.

   The Software License Agreement screen appears.

9. If you agree with the Software License Agreement, click the **Yes** button.

   The Information screen appears. Read the Information screen. You must have 5600 CNC Mill firmware version of at least 3.02.01.

10. Click the **Next** button.

    The Choose Destination Location screen appears. The default Destination Folder path shows C:\...\Lab-Volt Systems\Level 4 CNC. We suggest that you keep the designated folder.

    If require a different path, click the **Browse** button to select another folder

11. Click the **Next** button to select your Destination Folder path.

    The Setup Type screen is appears. As you click on each section a description of the setup type appears.

12. Click on the **Mill** setup selection, and then click the **Next** button.

    The Select Program Folder screen appears. A new folder name or a folder from the existing folder list may be selected. We suggest that you keep the default folder, **Lab-Volt Apps**.

13. Click the **Next** button.

    The Start Copying Files screen appears. Review the settings. If they are not correct, click the **Back** button to return to the previous screen to make corrections and then continue.

14. When you are satisfied with the Current Setting sections in the Start Copying Files screen, click the **Next** button.

    Setup copies the *Mill Level 4* software files to your computer, and then the Setup Complete screen appears.

15. Click the **Finish** button to complete the setup.

16. Close the Control Panel window, or if the installation was done through the Level 4 CNC Setup screen, just click on **Exit** to return to the Windows desktop, and skip step 17.

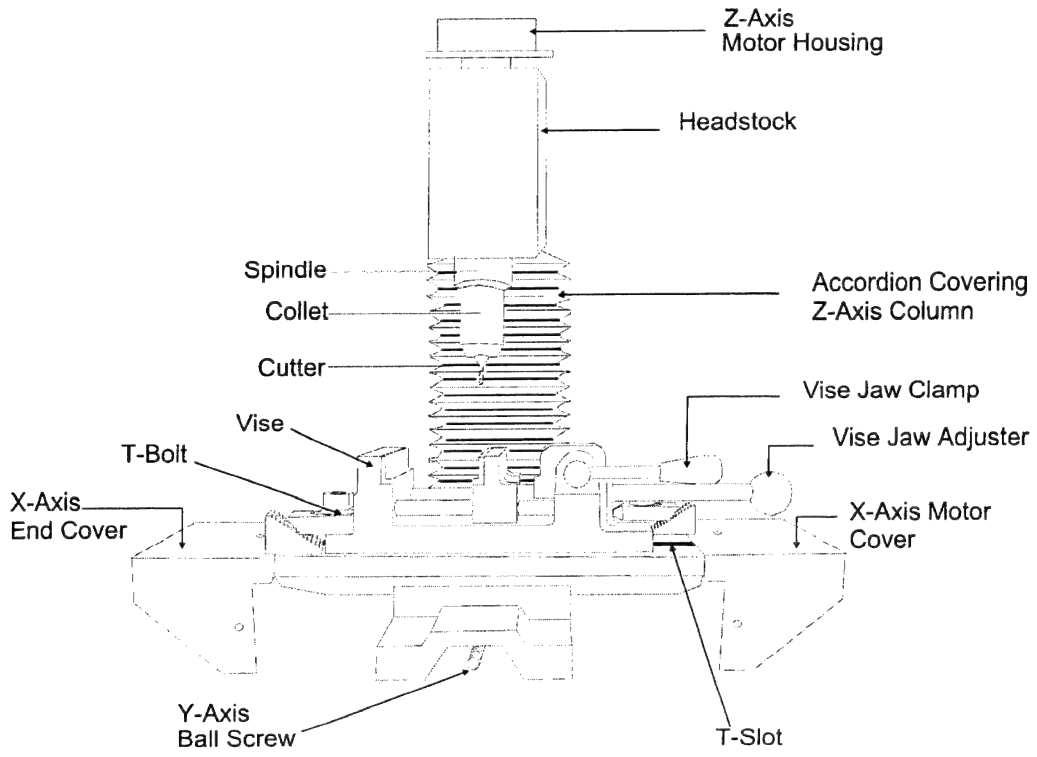
17. Close the My Computer window.

OPERATING THE 5600 CNC MILL

Functional Description of 5600 CNC MILL

The Lab-Volt Automation 5600 CNC Mill is in a cabinet that protects you during a milling operation. The 5600 CNC Mill is a fully functional mill that can be programmed from a computer.

The 5600 CNC Mill is a heavy-duty, fully functional mill that can be programmed from a computer. A mill cuts both vertically (like a drill) and horizontally. The 5600 CNC Mill uses stepper motors to move the milling table along the X-axis (left and right) and the Y-axis (backward and forward). An additional stepper motor moves the headstock along the Z-axis (up and down) and a 3/4 hp dc motor drives the spindle.

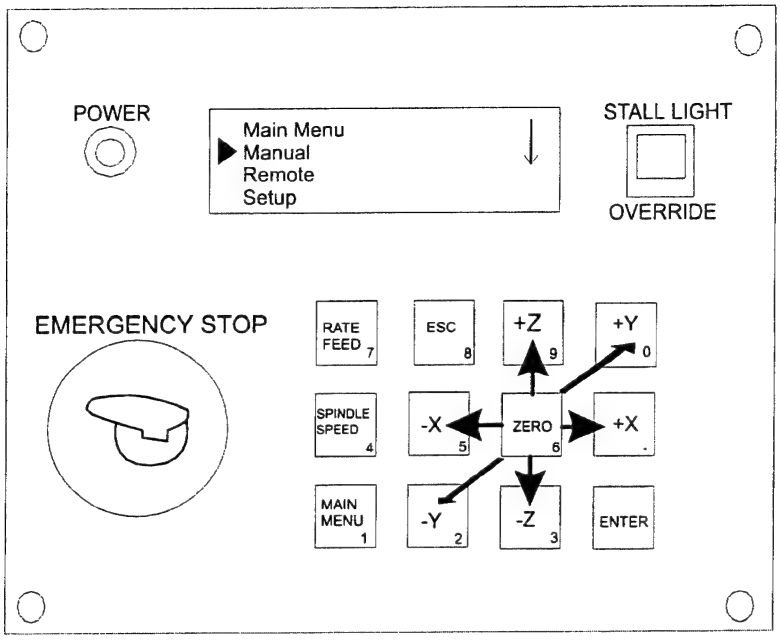


5600 CNC Mill Part Terminology

Stock to be milled can either be mounted directly on the mill table or can be secured in a vise. The CNC Mill accepts EIA-274-D standard G and M codes for mill operations. The codes can be used to create a part program, which can then be downloaded from the computer to the mill. Optionally, the mill can be operated manually using the mill control panel.

Control Panel

The control panel is located on the lower left front panel of the mill.



Control Panel

Control Panel

The following lights and keys are located on the control panel. The small numbers next to the keys described below are used to enter numerical values when required.

POWER LED – Indicates whether or not the power is on.

EMERGENCY STOP button – Controls the power to the spindle motor and to the stepper motors that move the milling table along the X and Y-axes and the headstock along the Z-axis. The power to the motors is cut off when the red *EMERGENCY STOP* button is pressed. It can only be released by inserting and turning the *EMERGENCY STOP* key.

FEED RATE$_7$ – Can be used to manually set the speed at which the mill table moves. An entry here overrides a computer-programmed feed rate. If an override occurs, an "F" is displayed in the lower right corner of the mill control panel screen.

SPINDLE SPEED$_4$ – Can be used to manually set the speed at which the mill cutter turns. An entry here overrides a computer-programmed spindle speed. If an override occurs, an "S" is displayed in the lower right corner of the mill control panel screen.

MAIN MENU1 – Pressing this key from the Remote menu returns the screen to the Main Menu. Pressing this key from the Manual or Setup menu accesses the Accessories menu.

ESC8 – Exits the current menu or aborts a control panel operation.

-X5 – Moves the mill table to the right along the X-axis.

-Y2 – Moves the mill table to the back along the Y-axis.

+Z9 – Raises the headstock away from the mill table.

Zero6 – Sets the reference point from which the program makes all moves. From the Manual menu, pressing *Zero* followed by an axis direction key accesses the Zero Menu.

-Z3 – Lowers the headstock towards the mill table.

+Y0 – Moves the mill table to the front along the Y-axis.

+X – Moves the mill table to the left along the X-axis.

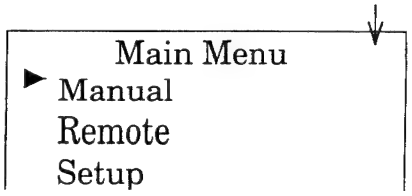
ENTER – Selects the menu option displayed, saves any changes made, or initiates implementation of the action displayed on the screen.

STALL LIGHT OVERRIDE button – Signals the system to stop all motors. They can be restarted (after any error conditions have been cleared) by pressing the *STALL LIGHT OVERRIDE* again. Refer to the **EMERGENCY PROCEDURES** section for more details.

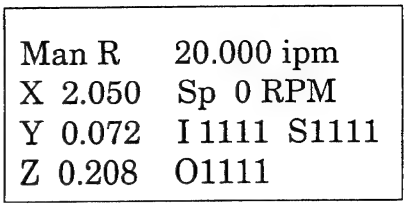
Menus

Main Menu

The Main Menu is the window for the built-in software that controls the lathe. There are five selections that can be made from the Main Menu: Manual, Remote, Setup, Accessories and Output. Use the *Z* keys to position the cursor next to a menu selection and press the *ENTER* key to select it.

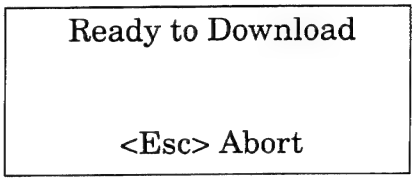


Manual Menu



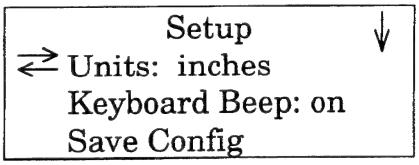
The Manual menu displays the feed rate of the mill table, the X and Y coordinates of the table, the Z coordinate of the headstock which holds the cutter, the current spindle speed, the Transistor Transistor Logic (TTL) Input/Output, and solenoid status. Use this menu to operate the mill in manual mode, to manually adjust the position of the table and the headstock. The *X, Y,* and *Z* keys are only operative from the Manual menu. Use the *ESC* key to return to the Main Menu.

Remote



The Remote selection is used to prepare the mill to receive instructions (e.g., a Level 4 part program) from an outside source. Select Remote to accept a part program download from the computer. Use the *ESC* key to return to the Main Menu.

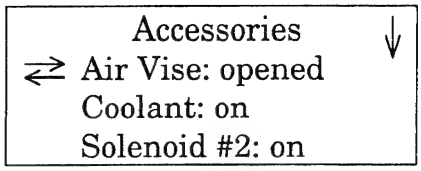
Setup Menu



The Setup menu contains selections for distance and feed units and for keyboard beep. It allows the operator to change the units of the mill table feed (inches or millimeters). Also, the Setup menu can be used to activate or deactivate a beep emitted when a key on the control panel keyboard is pressed and acknowledged. Use the *Z* keys to position the cursor next to a menu selection. Use the *X* keys to toggle the choice. Press the *Enter* key to save the change and return to the Main Menu. Changes made in this way will remain until they are changed again or until the mill is turned off. To save configuration changes so that they are used the next time the mill is started, move the cursor to Save Config and press *Enter*. To revert to the default configuration, select Reset Config and press *Enter*. Use the *Esc* key to return to the Main Menu without saving any changes on this menu.

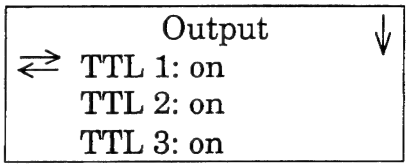
CAUTION: Factory set backlash settings are lost when the configuration is reset with Reset Config.

Accessories Menu



The Accessories menu can be used to manually change the status of the optional pneumatic vise, the coolant, and solenoids #2 and #3. Use the *Z* keys to position the cursor next to a menu selection. Use the *X* keys to toggle the choice on the current line. To save changes made on this screen and return to the Main Menu, press *Enter*. To return to the Main Menu without saving changes, press *Esc*.

Output Menu



The Output menu can be used to manually change the status of the four Transistor Transistor Logic (TTL) outputs. The TTL I/O communications capability can be used with up to four devices. Each TTL output can be set to "on" or "off". Use the *Z* keys to position the cursor next to a menu selection. Use the *-X* or *+X* to toggle the choice. To save changes made on this screen and return to the Main Menu, press *Enter*. To return to the Main Menu without saving changes, press *Esc*.

Safety Interrupts

The 5600 CNC Mill is shielded with clear Lexan® doors and is provided with several safety interrupts to ensure safe operation.

Auto Cutoff Doors

The two Lexan doors on the front of the mill are each fitted with a magnetic interlock to prevent operation of the mill when either door is not properly closed. If either door is opened during mill operation, all motors are turned off immediately.

Emergency Stop Button

This button shuts off the power to all motors. After pushing this button, the system can only be restarted by using the Emergency Stop button key.

Stall Light Override Button

This button interrupts the implementation of a part program by signaling the system to stop all motors. The program can be restarted by pressing the *Stall Light Override* again and clearing any error conditions.

Limit Switches

There are switches on each of the tracks that prevent the mill table and the headstock from moving past their limits. If a switch is triggered causing a lockup, return to the Main Menu, select Manual and press *ZERO*, *Esc*. Use the appropriate *X, Y,* or *Z* key to move the axis away from the limit. If the axis is still jammed, refer to **Appendix A** for further instructions. Reset the switch(es) by pressing *Esc*. This also returns the screen to the Main Menu.

If any of these buttons or switches are triggered, an error message is displayed with the option to continue or abort operations. Correct the error condition before attempting to resume operations. Refer to ***Emergency Procedures*** for more details.

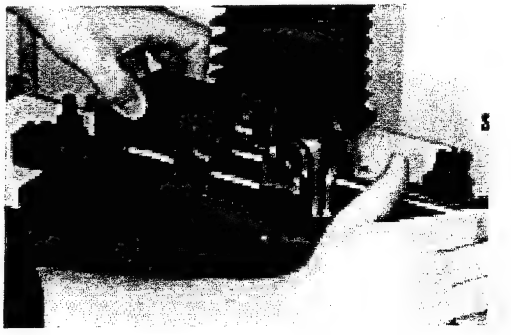
Startup of the 5600 CNC Mill

CAUTION: Do not turn on the power to the 5600 CNC Mill until you are instructed to do so. Put on your safety goggles to protect your eyes. Clean and remove any filings and scraps of stock from a previous milling operation that may be in the 5600 CNC Mill cabinet. **Good housekeeping prevents accidents.**

Load the Stock

Obtain a piece of stock. You were provided with 100 blocks of 2" x 2" x 0.5" of machinable wax stock.

1. Pull the black-knobbed rod to open the movable jaw of the vise to the size needed to accommodate the block.
2. Place a wood shim of the same or slightly smaller width than the block to be milled (width referring to the dimension being clamped by the vise) onto the inside ledges of the vise.
3. Lay the block to be milled on top of the shim.
4. Line the block up flush with the front of the vise jaws.
5. Use the black-knobbed rod to push the movable jaw of the vise closed.
6. Press the vise clamping lever clockwise all the way to the right to clamp the block in place. This lever should be parallel to the black knobbed rod when in the clamp position.
7. Close the mill doors.



Vise on the Mill table

Turn Power On to the 5600 CNC Mill

1. Press the power switch located on the back panel of the 5600 CNC Mill to the *ON* position (depress the side with the "-" symbol).

   The *Power* LED lights up. Wait for about 60 seconds for the mill to initialize. A title screen appears on the control panel followed by the mill **Main Menu** screen.

Tool Movement Rates

Before downloading the part program for machining, the programmed reference zero (PRZ) needs to be set on the mill stock. This step must be done before machining any part.

Before you can set the PRZ, you must become skilled at manually moving the end mill tool or cutting tool.

1. On the 5600 CNC Mill control panel display place the pointer next to **Manual** by using the *-Z* and *+Z* keys.

2. Press the *Enter* key on the mill control panel.

   The **Manual** screen is displayed on the control panel. The X, Y, and Z values on your screen may be different.

Man R	14.000 ipm
X 2.050	Sp 0 RPM
Y 0.072	I 1111 S1111
Z 0.208	O1111

3. Press the *X, Y,* and *Z,* + and - keys to practice moving the end mill.

   Move the end mill around a little to get the feel of how it moves. Notice that the mill travels at the default continuous rapid rate of 14 ipm.

   The line MAN R 14.000 ipm at the top of the mill's display means that the mill is currently set to travel, in the manual mode, at the continuous rapid rate of 14 inches per minute.

4. Press the *Enter* key on the mill control panel once.

   The same line now reads MAN F 10.00 ipm. This changes the continuous travel rate from the rapid rate of 14 inches per minute to 10 inches per minute.

   It may be easier to position the end mill at the feed rate of 10 ipm instead of 14 ipm.

5. Move the end mill again with the *X, Y,* and *Z,* + and - keys and notice that the rate is slower.

Incremental Tool Distances

The manual mode also offers incremental distances as choices for tool movement. These incremental distance moves are performed at the default feed rate of 10 ipm or at a feed rate that you may set which will be shown on the control panel display.

> **WARNING:** When the end mill is set to move in incremental steps, the X, Y, and Z limit switches are disabled.
>
> When in the incremental mode, be very careful not to hit the stock with the end mill. Damage could occur to the 5600 CNC Mill.
>
> You are shown how to use the incremental mode to set the PRZ in the following steps.

1. Press the *Enter* key on the control panel once more.

   The display now shows 0.001 F 10.00 ipm on the first line. This means that one click of any of the axis keys will move the tool only 0.001 at a rate of 10.00 ipm.

2. Press the *Enter* key on the mill control panel four more times.

   The display should now show 0.100 F 10.00 ipm. The mill tool will move 0.100" each time an axis key is pressed.

3. Press the +*Z* key once.

   The mill moves up exactly 0.1 inch. The travel options in the manual mode are flexible to provide the operator with accurate tool placement capability.

4. When the **Manual** screen is displayed, the *Enter* key on the control panel is used to move between the rapid rate, feed rate and the preset incremental distances.

   Once the last incremental option is reached, the list starts again at the top with MAN R 14.00 ipm, which is the rapid rate.

   The following table shows the possible preset tool movements that are available.

Top Display Line	Movement
Man R 14.000 ipm	Rapid, 14 inches per minute
Man F 10.000 ipm	Feed rate, 10 inches per minute
0.001 F 10.000 ipm	Move one thousandth of an inch/click
0.005 F 10.000 ipm	Move five thousandths of an inch/click
0.010 F 10.000 ipm	Move one hundredth of an inch/click
0.050 F 10.000 ipm	Move five hundredths of an inch/click
0.100 F 10.000 ipm	Move one tenth of an inch/click
0.500 F 10.000 ipm	Move five tenths of an inch/click
1.000 F 10.000 ipm	Move one inch/click

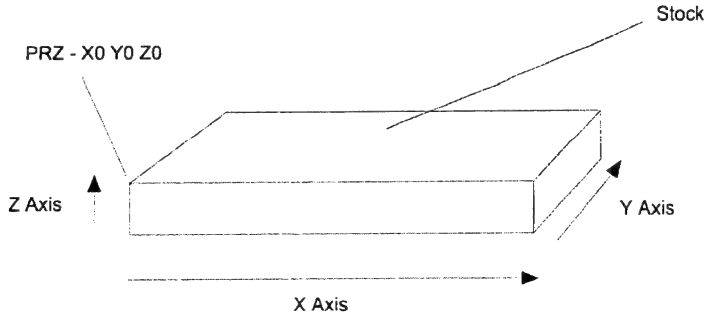
5. Continue to experiment with moving the mill tool manually at different rates and incremental distances until you are familiar with the procedure for moving the tool.

6. Return to the feed rate setting for manual travel (MAN F 10.00 ipm).

Position the End mill at the PRZ

The PRZ is the point from which all absolute movements are measured. Therefore, an incorrect PRZ causes the entire program to be off.

The recommended PRZ for the 5600 CNC Mill is X0 Y0 Z0. This point is the front left corner of the stock just touching the surface.



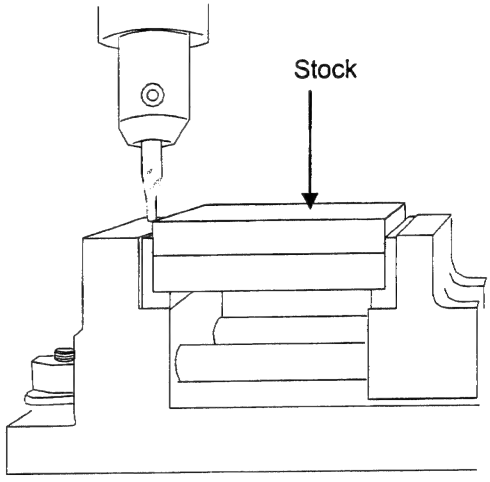
Programmed Reference Zero (PRZ)

It is important that the center of the end mill be centered over the corner, not inscribed in the corner. If the PRZ is set with the end mill inscribed in the corner instead of centered over the corner, the design will be off position by one half the diameter of the end mill. The PRZ must represent the center of the end mill, not the edge of the end mill as shown in the following figure.

Proper Position for End Mill at PRZ

1. With the feed rate set at MAN F 10.00 ipm, manually position the end mill near the PRZ on the stock.

2. Use the smaller incremental distance settings to exactly position the end mill at the PRZ. When you are very close to the PRZ, a setting of about 0.005 F 10.000 ipm should be used to exactly position the end mill.



Stock in Vise

Set the PRZ

Now that the end mill is positioned at the correct PRZ, a value of 0 needs to be assigned to each axis. This can either be done one axis at a time, or all three can be assigned a zero value simultaneously.

1. Press the *ZERO* key (center key) on the mill control panel once, then press either of the *X* keys.

   The display now shows the mill waiting for a new X value to be entered.

2. Press the *0* key (upper right) on the mill control panel, then press *ENTER*.

   This sets the X-axis to zero. This procedure can be used to assign a new value to any individual axis. Do not assign a 0 to the Y and Z yet. The next step shows how to zero all three axes simultaneously.

   The Lab-Volt 5600 CNC Mill allows you to conveniently zero all three axes simultaneously.

3. Press the *ZERO* key in the center twice.

   This sets all three axes to zero.

   After the end mill is in the PRZ position, and all axes have been assigned a value of zero, the mill is ready to have the part program downloaded from the computer.

4. Press *ESC* on the mill control panel to return to the **Main Menu**.

Prepare for Part Program Download

1. Use the *Z* keys to move the cursor to **Remote**.

Ready to Download
<Esc> Abort

2. Press *ENTER* to select the **Remote** screen.

   The mill is now ready to accept a program from the computer.

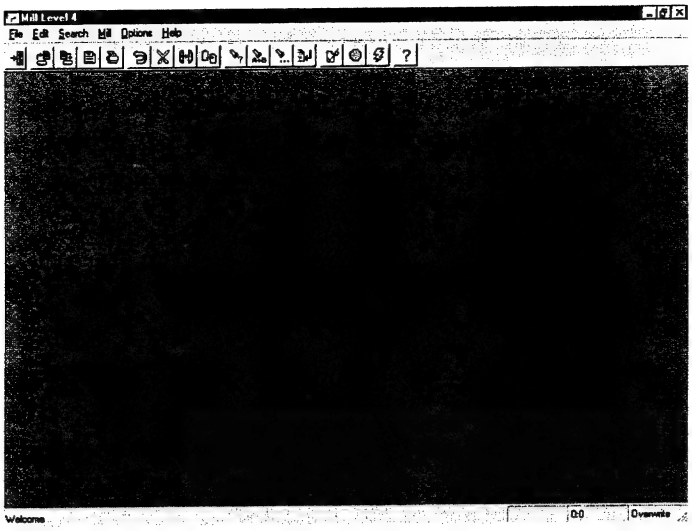
Mill Level 4 Software

Start the Mill Level 4 Software

1. From the Windows 95, 98, on NT 4.0 Desktop, start the *Mill Level 4* software by, clicking on the **Start** button (lower left), then point to **Programs**, then **Lab-Volt Apps**, and click on **Mill Level 4.**

   Or if you have a **Mill Level 4** icon on your Desktop, you can double click on it to open the *Mill Level 4* software.

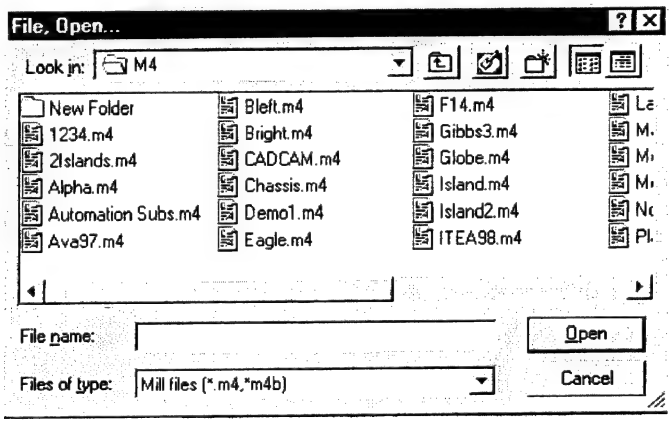
The Mill Level 4 main screen opens.



Level 4 Mill Main Screen

2. On the Menu Bar of the Mill Level 4 screen, click **File**, then **Open**.

   The File, Open window appears with a list of the mill programs stored in the M4 folder.



File Save Menu

3. Click **Cancel** to close the File, Open window.

4. On the Tool Bar of the Mill Level 4 screen, click the **Open** button  .

   The File, Open window appears again with the same listing of programs.

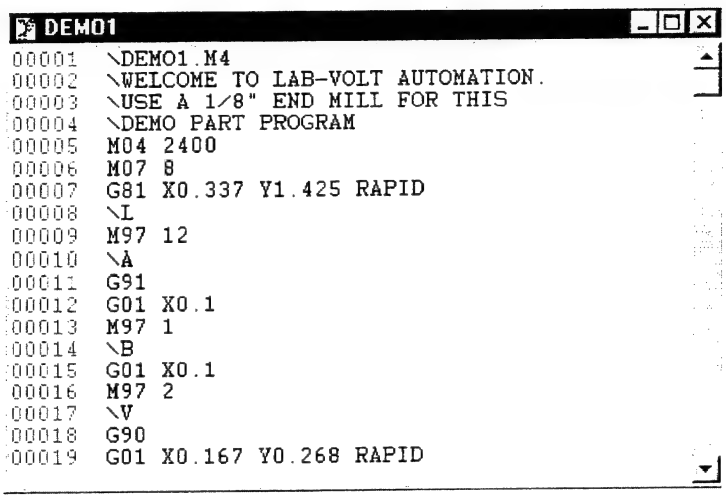
   The Tool Bar buttons on the Mill Level 4 screen perform the same operation as the Menu Bar equivalents.

5. In the File, Open window, click on Demo1.m4.

   The Demo1.m4 file name becomes highlighted.

6. Click on **Open**. (An alternate way to open the Demo1 editor window is to double click on Demo 1.m4 in the File, Open window.)

   The Demo1 editor window opens.



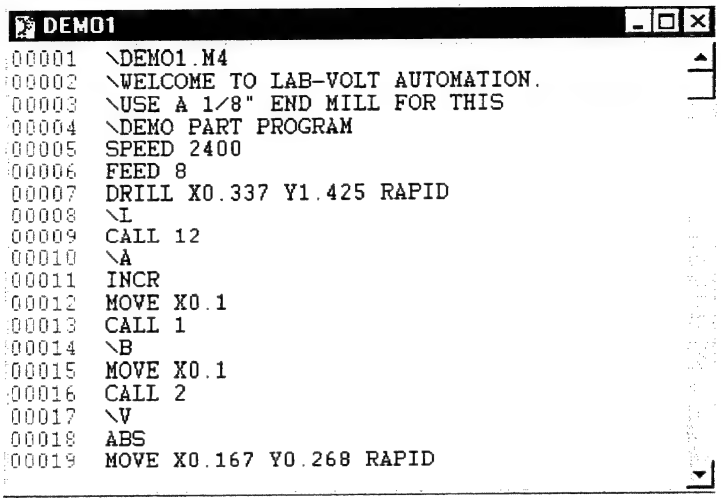
DEMO1 Editor Window with ISO Codes

The editor window contains a list of the steps, or lines, which make up the part program. This program was written using ISO (Industrial Standard Operation) machine codes. For example, G91 indicates incremental mode and M97 calls a subroutine.

The program editor in Level 4 allows you to toggle between the G and M codes and their equivalent conversational codes.

7. On the Menu Bar, click **Edit** and then **Conversational**.

   The program listing in the editor window changes to the conversational codes as shown in the following figure.



DEMO1 Editor Window with Conversation Codes

8. Click anywhere in the program, so that a vertical flashing cursor appears somewhere in the text.

   If you wanted to, you could edit text at the flashing cursor.

9. Press the *F2* key on the top row of the keyboard.

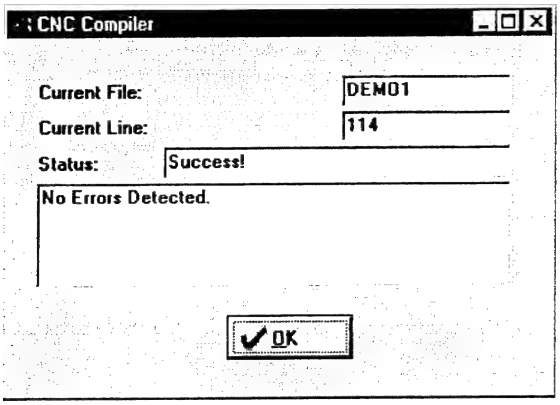
   The program listing changes back to ISO codes. Leave it in the ISO code mode.

   You can easily change between G and M codes and conversational codes by either choosing **Conversational** from the **Edit** menu or pressing the *F2* key.

Compile a Part Program

1. On the Menu Bar, click **Mill** and then **Compile**.

   The CNC Compiler window appears.



Compiler Menu

   The CNC Compiler window shows the Current File name, the Current Line, which is the line number after the last program line, the Status and comments on any errors.

   Compiling a program tests the program for errors and verifies it for emulation and downloading to the mill. All part programs must be compiled before they can be emulated or downloaded. There should be no errors in the DEMO1 part program.

2. Click **OK** in the CNC Compiler window to close the window.

3. On the Tool Bar, click the **Compiler** button  .

   The CNC Compiler window opens again.

4. Click **OK** to close it.

   You can choose to use the **Compiler** button or use the Menu Bar equivalent to open the CNC Compiler window.

Emulate a Part Program

NOTE: If you compile a part program with the Mill 3D Emulator window open, the emulation will not run. You must have the Mill 3D Emulator window closed when you compile a program.

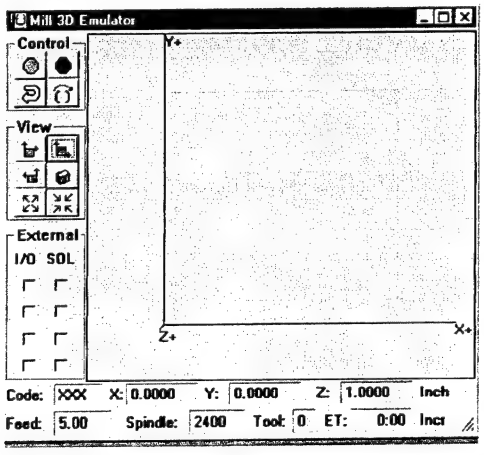
Once a part program has been successfully compiled, you can run an emulation. When you emulate a program, the computer performs a graphic simulation of the open part program.

All part programs must be emulated from several views before actual production. This ensures that the machining will occur without errors or tool crashes. Simulation of the milling of the part before production saves costly damage to the mill and wasted stock due to errors.

1. Click on the green **Emulator** button  on the Tool Bar.

   The Mill 3D Emulator window (shown below) appears.

   If the current part program was not compiled when the **Emulator** button is clicked, Level 4 software compiles it automatically before the Mill 3D Emulator window opens. The window opens after you click on **OK** in the CNC Compiler window.



Mill 3D Emulator Window

You can also open the Mill 3D Emulator window by clicking **Mill** on the Menu Bar and then **Emulator.**

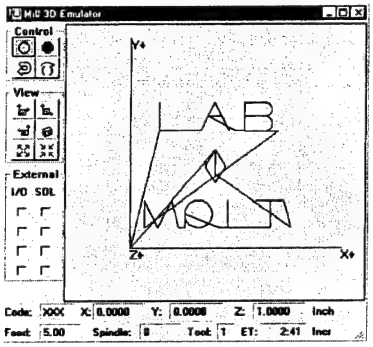
When the Mill 3D Emulator window appears, the stock is shown in the top view  .

The top view emulation shows the position of your design on top surface of the stock. Running the top view emulation before machining ensures you that the designed part is going to fit on the stock, without extending beyond any of the edges.

To ensure that the vise will not be damaged during the cutting of the part, it is important to perform a top view emulation for every part to be cut on the Lab-Volt 5600 CNC Mill.

2. Click on the green **Start** button  under Control on the Mill 3D Emulator window.

   You will see the following top view graphic of the milling emulation.



Emulation Window

In this view, the PRZ is in the lower left corner. Notice that some of the lines indicating the tool path are red and others are blue. The red lines represent movement at the rapid rate with the end mill out of the stock and the blue lines represent travel at the cutting feed rate.

3. Click the **Reset** button  .

   This clears the emulation screen and prepares for a new emulation.

View Control Buttons

You have following six view buttons available under View on the Mill 3D Emulator window:

 **Front View** button

 **Top View** button

 **Side View** button

 **3D View** button

 **Zoom In** button

 **Zoom Out** button

1. Click the **Side View** button  .

   The side view emulation allows you to confirm the cut depth of a part program. You can determine if the end mill is going to cut at the correct depth and whether or not the tool will go all the way through the stock at any point in the program.

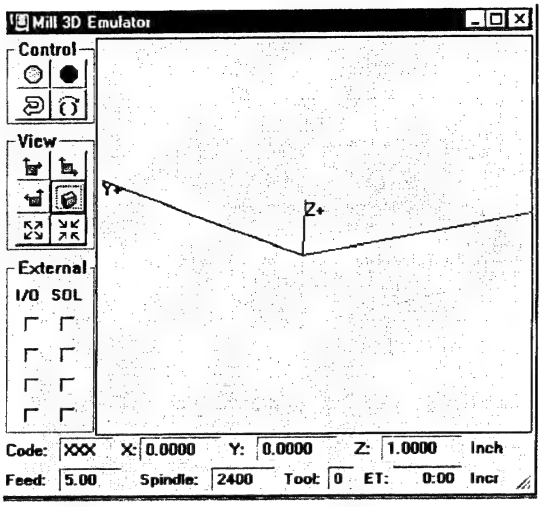
2. Click on the green **Start** button  .

   The emulation is again performed with different colored lines; blue for end mill feed commands and red for rapid commands.

   The end mill's depth into the stock can easily be seen from this view. This part program does not cause the tool to cut all the way through the stock. The front view can also be used in the same way to easily check the depth of cut.

   The 3D emulation shows the part in perspective. Your image may not be centered or may appear distorted from certain 3D views.

3. Click the **3D View** button  .



Emulation Window

4. Click on the **Start** button  .

   The 3 dimensional emulation view allows you to view a simulated representation of a part program showing all three axes simultaneously.

5. Click three times on the **Zoom In** button  .

   You now have a closer view of the stock. This can be helpful when viewing fine detail. The arrows on the **Zoom In** button indicate that the view of the part gets larger.

6. Click the **Zoom Out** button  three times. The original view is restored. The arrows on the **Zoom Out** button indicate that the view of the part gets smaller.

7. Click the **Zoom Out** button three more times.

   The new view of the stock is from farther away. This can be helpful when viewing large pieces.

8. Click the **Zoom In** button three times. The original view is restored.

   All of the view manipulation used so far has been from the default camera position. *Mill Level 4* software also allows you to change the camera's position within any given view.

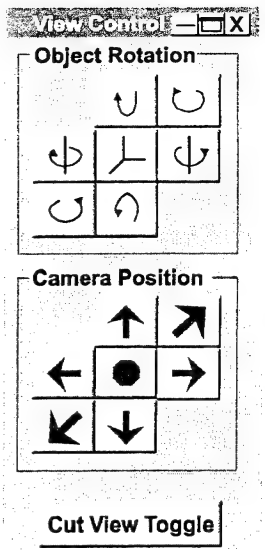
View Control Window

1. Click **Mill** on the Menu Bar and then click **View Control**.

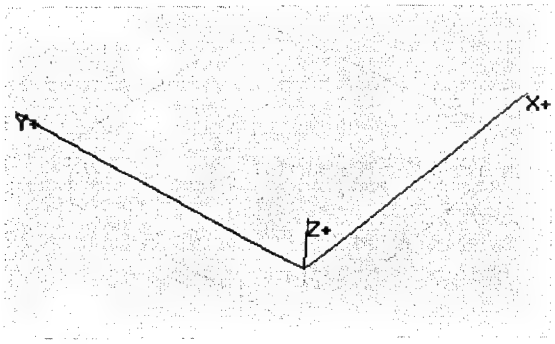
   The View Control window is displayed. Point in the blue header at the top of the View Control window and hold the mouse button down. Now drag the View Control window off of the Mill 3D Emulator window. This gives a clear view of both windows instead of one window being on top of the other.

2. Click the Object Rotation and Camera Position buttons on the View Control window to change the angle and rotation of the view.

3. Experiment with the buttons and tilt the piece to get a little better view of the top as shown in the following figure.



Veiw Control Window



Emulation Screen Capture

4. Click the **Start** button.

   The emulation occurs in the new custom view. The View Control option allows you to adjust for difficult viewing conditions.

   Notice how some of the red, rapid end mill, path lines obstruct the view of the blue end mill feed rate lines. In this program, all of the end mill movements performed at the feed rate (blue lines) are going into or cutting the stock. That is, the movements performed at the cutting feed rate have a negative Z value.

   The **Cut View Toggle** button on the View Control window allows you to perform an emulation that shows only the end mill movements with negative Z values.

5. Click the **Reset** button in the Mill 3D Emulator window to clear the last emulation.

6. Click the **Cut View Toggle** button in the View Control window.

7. Click the **Start** button in the Mill 3D Emulator window to run the emulation again.

   With the **Cut View Toggle** button depressed, only the blue cut paths that contain negative Z values are displayed along with red vertical lines that indicate where the end mill entered the stock.

   This view feature is very useful for intricate designs where too many path lines crowd the emulation. It allows you to focus on only the paths where the end mill is cutting stock.

8. Click the **Reset** button to clear the emulation.

9. Click the **Cut View Toggle** button in the View Control window to turn off the cut view mode.

10. Click the **Close** button in the upper right corner of the View Control window to close the View Control window.

Step Operation Emulation

Mill Level 4 software allows you to step through a part program line by line so you can easily identify at what point in the program that each action occurs. Stepping through a program in this manner can also be useful when searching for errors. You use the Trace button to emulate one command of the part program at a time.

1. Click the **Trace** button once to move to the first command of the part program. Continue to click the **Trace** button until the entire image is traced.

   Notice that as you click for each new step, the program step you trace is highlighted in the program editor at the left. When you get to the end of the program, the next click of the **Trace** button will return you to the start of the program.

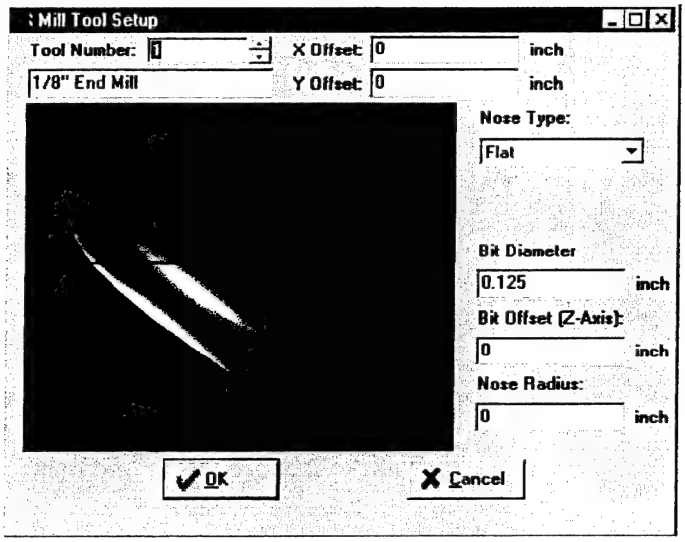
2. Click the **Close** button in the upper right corner of the Mill 3D Emulator window to close the emulator.

Check the Tool Library

It is especially important to confirm that the cutter diameter is correct to ensure the highest degree of milling quality. If any information is incorrect, enter the correct data.

1. Click on **Options** in the Menu Bar, and then click **Tool Setup**.

   The Mill Tool Setup window appears.



Mill Tool Setup Window

   Verify the following information:

   - The Tool Number is 1, and the end mill is a 1/8-inch End Mill.
   - The X and Y Offsets are 0 inches.
   - The Nose Type is Flat.
   - The Bit Diameter is 0.125 inch, the Bit Offset (Z-axis) is 0 inch, and the Nose Radius is 0 inch.

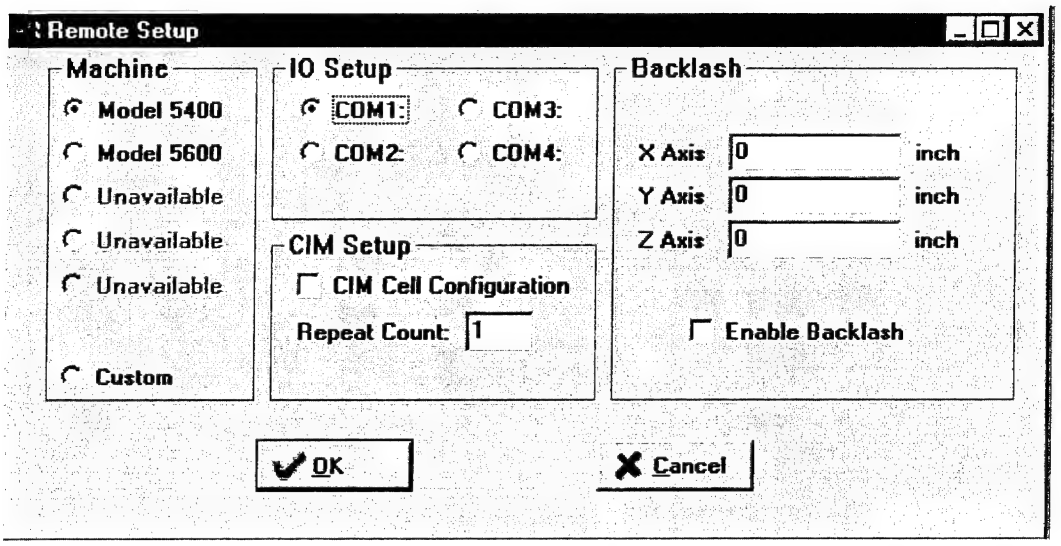
2. When finished with the screen, click **OK**.

Download Part Program and Mill the Part

Before downloading, you should check the communications settings in *Mill Level 4* software.

1. On the Menu Bar of the Mill Level 4 screen, click **Options**, then **Communications**, and then **Remote Setup**.

   The following Remote Setup window appears.



Remote Setup Window

Look at the Machine and IO Setup fields. Make sure the machine checked is Model 5600 and the appropriate COM port is selected.

The COM port is the computer port to which the mill serial cable is connected, usually COM1 or COM2. Ask your instructor, what COM port the 5600 CNC Mill connects to.

Backlash is the amount of play in the axis drive mechanism when the axis direction is changed. Backlash values are measured and preset at the factory for each machine. You do not have to make any entry under Backlash

The CIM Setup is for communications connections in a computer-integrated manufacturing (CIM) or flexible manufacturing system (FMS). You do not have to make any entry.

Once these settings are made, there is no need to recheck them unless the model or port is changed. Once Level 4 is set up to communicate with the mill, these settings remain until you change them.

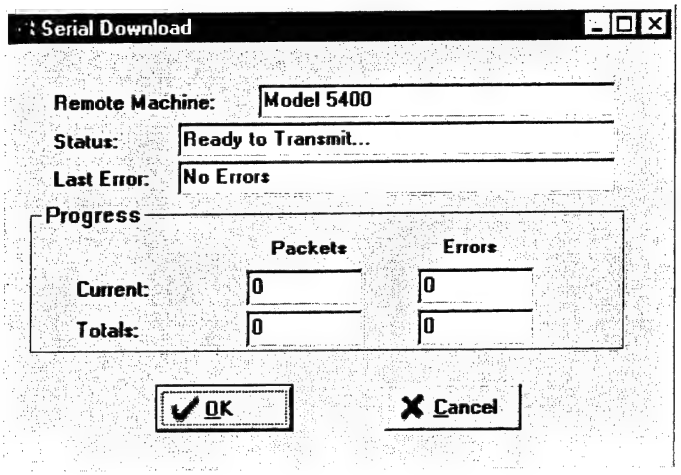
2. Click **OK** to accept settings in the Remote Setup Window.

   At the 5600 CNC Mill, check to be sure that the control panel displays Ready to Download. If not, select **Remote** on the **Main Menu** screen, and press Enter.

3. On the computer, click the **Download** button  .

   The same command is also available under **Mill** on the Menu Bar.

   This brings up the Serial Download window below with the Status box showing "Ready to Transmit".



Serial Download Window

Click **OK**.

At the 5600 CNC Mill, the control panel displays "Downloading" while the process is taking place.

Downloading
<Esc> Abort

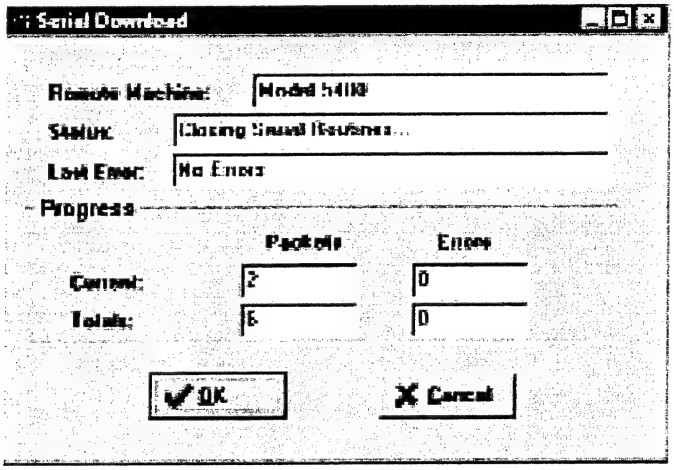
When downloading is finished the mill control panel displays "Downloading Complete".

Download Complete
<Esc> Abort

If you do not see "Downloading Complete" on the mill control panel, the Serial Download window will indicate an error in downloading. If necessary, recheck the settings in the Remote Setup window.

When downloading is complete, the Status box shows Closing Serial Routines. The Last Error box shows "No Errors".

4. When the 5600 CNC Mill control panel displays "Download Complete", close the Serial Download window on the computer by clicking **OK**.



Serial Download Window

CAUTION: Make sure the mill door is closed and the end mill and vise are clear of any obstructions. Make sure the vise wrench has been removed from the cabinet. Be sure that you are wearing safety goggles to protect your eyes.

5. On the mill control panel, press the *ENTER* key to mill the part.

   If it appears that the mill will crash or encounter trouble, press the *EMERGENCY STOP* button.

   Do not open the door while the mill is running or the program will be interrupted.

   When milling is complete, the mill returns to the Home Position. This is one inch above the PRZ or at a custom Home Position. You set custom Home Position in the Advanced Mill Options window of the *Mill Level 4* software.

6. On the mill control panel, press *ESC* to return to the **Main Menu**.

Shutdown the Mill

1. Turn off the power to the 5600 CNC Mill by pressing the power switch located on the back panel to the *OFF* position. Confirm that the *POWER* LED on the control panel is off.

2. Open the door and remove the milled part from the vise.

Shutdown the Computer

1. At the computer, click **File** on the Menu Bar.

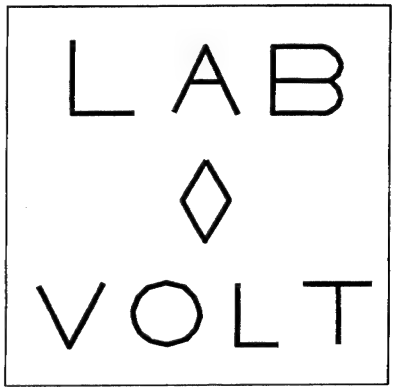
2. Click **Exit**.

3. If asked to save changes, select **No**.

Inspect the Milled Part and Clean the Mill

1. Inspect your milled part. It should appear as shown in the figure above.

2. Thoroughly clean the 5600 CNC Mill Cabinet of any stock scraps or filings before leaving the workstation area.



Milled Part

Load Existing Level 3 Part Programs

Level 4 Mill software is completely compatible with Level 3 part programs.

1. Use Windows Explorer to copy Level 3 part programs into the M4 folder of the Mill Level 4 software. The M4 folder normally is in the \Program Files\Lab-Volt Systems\Level 4 CNC folder.

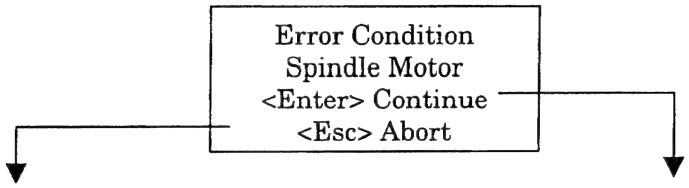
2. Or access the programs directly by clicking on **Open** in the Mill Level 4 **File** menu.

   Either way, change the file type to Mill Level 3 Files (*.m3) to access the Level 3 part programs.

Emergency Procedures

Stall Light Override Button

To interrupt the milling procedure because of a mechanical fault such as the end mill coming loose, an inadequately secured stock, impending damage to the mill, or an incorrect PRZ setting, press the *STALL LIGHT OVERRIDE* switch. This stops all milling operations immediately and an "Error Condition" message is displayed.



Press *ESC* on the control panel to abort the run.

Spindle Motor Correct Error Condition

or

Press *ESC* again to leave the table in its current position or press *ENTER* to return the cutter to its home position, which is one inch above the X, Y coordinates of the PRZ.

How to Return? <Enter> Go to Home <Esc> Stay Here

Correct the problem. In most cases the corrective action begins with:

Press the *STALL LIGHT OVERRIDE* then *ESC* to access the Main Menu. Use a *Z* key to move the cursor to Manual. Press *ENTER*. Using the *+Z* key, raise the tool so that the problem can be accessed. When the problem has been corrected, go to REMOTE and continue as described in **Download Part Program and Mill the Part**.

Ready to Download <Enter> Run Program <Esc> Abort

Press *ENTER* to continue.

Warning Resume Machining? <Enter> Continue <Esc> Abort

Press *ENTER* again to continue.

Spindle Motor Correct Error Condition

Press the *STALL LIGHT OVERRIDE* to return to the Manual menu and restart the program where it left off.

G81 F 8.000 ipm X 0.240 Sp 1000 RPM Y 0.100 I1111 S1111 Z 0.435 O1111

Emergency Stop Button

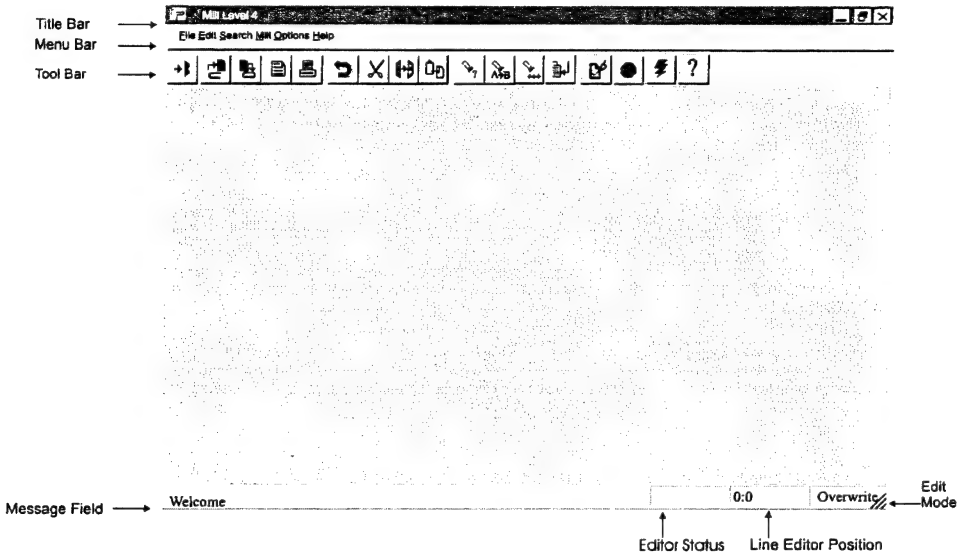
Press the *EMERGENCY STOP* button on the lathe control panel to stop all lathe motors immediately. When the problem has been corrected, use the key to unlock the *EMERGENCY STOP* button. Restart operations.

Starting Mill Level 4 Software

From the Windows 95, 98, on NT 4.0 Desktop, start the *Mill Level 4* software by, clicking on the **Start** button (lower left), then point to **Programs**, then **Lab-Volt Apps**, and click on **Mill Level 4**.

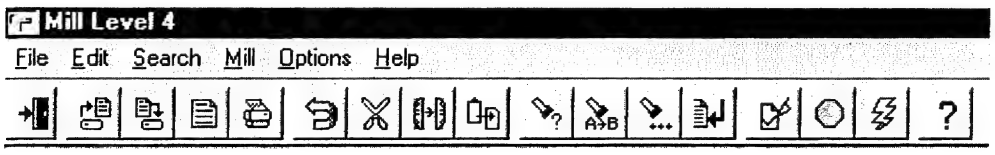
Or if you have a **Mill Level 4** icon on your Desktop, you can double click on it to open the *Mill Level 4* software.

The Mill Level 4 title screen appears briefly, and then the Mill Level 4 main screen opens.



Mill Level 4 Main Screen

The buttons on the Tool Bar show below offer "point and click" access to some of the most useful features in the *Level 4 Mill* software.



Mill Level 4 Toolbar

Record the Backlash Values

Record the factory-set backlash values in case they are accidentally lost during a configuration reset. Backlash is the amount of play in the axis drive mechanism when the axis direction is changed. This causes a discrepancy in the distance the axis travels when it changes direction. A backlash value is applied to each axis to compensate for the discrepancy. Backlash values are measured and preset at the factory for each machine. However, if the backlash values are lost, you can reenter them using the procedure, *How to Reset Backlash Values*, in the Mill Level 4 Software Documentation.

1. If necessary, turn on the power to the 5600 CNC Mill.

   The *POWER* LED lights up. Wait for about 60 seconds for the mill to initialize. A title screen appears on the control panel followed by the mill's **Main Menu**.

2. On the 5600 CNC Mill control panel display, place the arrow next to **Remote** by using the *-Z* and *+Z* keys.

3. On the mill's control panel, press *Enter*.

4. On your personal computer that connects to the 5600 CNC Mill, click on **Mill** in the Menu Bar on the Mill Level 4 screen.

5. On the **Mill** menu, click on **Query Remote**.

   The Remote Query window appears.

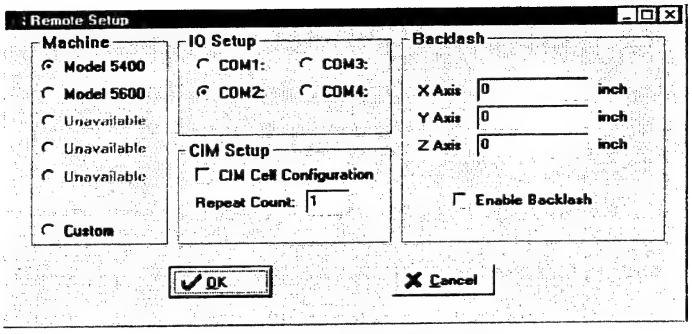
6. Click on **Detect** to test the connection between your computer and the mill to retrieve configuration information about the mill.

   After a successful query (the connection is good), the backlash values for the connected mill will be displayed in the Remote Setup screen, which you will access in Step 8.

7. Click on **Close.**

8. To view the Remote Setup screen, click on **Options** in the Menu Bar, then click on **Communications**, and then click on **Remote Setup**.

   The Remote Setup window appears and displays the backlash values for each axis of the mill.



Remote Setup Window

9. Record the backlash values for your 5600 CNC Mill.

   Backlash Values

   X Axis _____

   Y Axis _____

   Z Axis _____

CHANGING AN END MILL

The 5600 CNC Mill comes with a factory-installed 1/8-inch end mill. The end mill is held in the mill's spindle by a tapered collet that is screwed into to a draw bar that fits through the spindle. When you tighten the hex nut on the draw bar, the tapered collet compresses on the shaft of the end mill to hold the end mill tightly in the spindle.

A setscrew end mill holder that screws onto the spindle holds some end mills in place. With a setscrew end mill holder, the draw bar and tapered collet are not necessary.

The following procedures are for the following:

- Removing an End Mill and Tapered Collet
- Installing an End Mill and Tapered Collet
- Installing an End Mill and a setscrew end mill holder.
- Removing an End Mill and a setscrew end mill holder.

You do not have to perform the above procedures until it is necessary to change an end mill.

In Activities 2 through 7 in the Student Manual, you use the 1/8-inch end mill and a tapered collet that is factory installed on the mill.

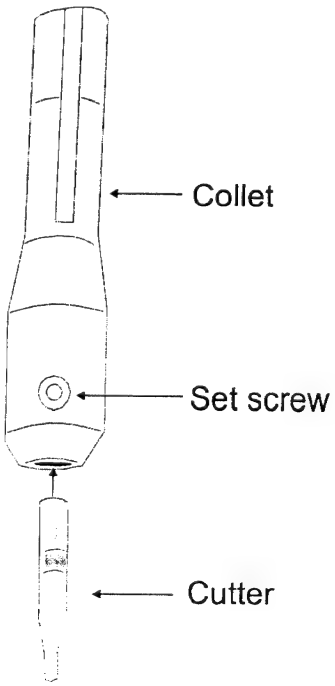
Change the Cutter

The DEMO 1 part program is designed for use with the 1/8-inch end mill cutter, which has been factory installed. To verify that the correct cutter is installed or to change the cutter, use this procedure.

INSTALL THE CUTTER

If the replacement cutter has the same shaft diameter as the current cutter:

1. Use the 3/16-inch wrench provided to loosen the setscrew in the collet.
2. Remove the cutter to be replaced.
3. Make sure the setscrew is out far enough for the new cutter to be inserted. Insert the cutter so that the flat portion of the cutter shaft faces the setscrew.. While holding the cutter in place, tighten the setscrew with the 3/16-inch hex wrench to lock it in place.

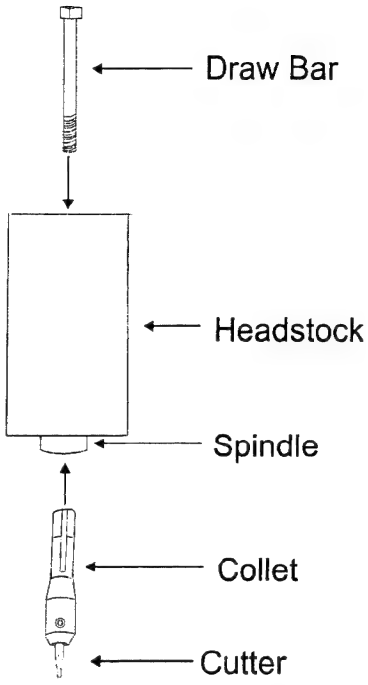


Collet

If the cutter has a shaft diameter different from the current cutter, the collet must be changed before the new bit can be installed. Refer to **Change the Collet**, then install the cutter as described above.

Change the Collet

Additional collets have been provided to accommodate different cutter shaft sizes. This procedure describes how to install a collet into the spindle.



Headstock and Collet

CHANGE THE COLLET

1. Select Manual from the Main Menu. If the cursor is not next to Manual, use the *Z* keys to move the cursor to it. Press *Enter* to select Manual.

2. Move the headstock to the uppermost position using the +*Z* button.

3. Place the 3/16-inch hex wrench in the hole on the side of the collet to keep it from turning.

4. Loosen the draw bar nut at the top of the headstock, turning it counterclockwise with the 5/8-inch wrench until the threads are no longer engaged and the bar can be lifted freely.

5. Tap the head of the drawbar to release the collet.

6. Insert the new collet in the spindle and tighten the draw bar nut by turning it clockwise.

WARNING: The end mill is very sharp. Be careful when working with it or near it.

Routine Maintenance

The only routine maintenance required is to clean and lubricate the equipment. After use, remove the vise and clear away all debris using the chip brush or a small vacuum. Clean off the X and Y-axes of the milling table and the Z-axis of the headstock.

The mill should be frequently lubricated with light machine oil such as 3-IN-ONE® oil. Put a couple of drops of oil on the Y-axis table and spread it around to thinly cover the sliding top surface. Lightly oil the front surface tracks and the side rails of the Z-axis. Put one drop of oil on each of the three lead screws (the long screws that the axes move along). The lead screw for the X-axis is underneath the axis and the lead screw for the Z-axis is behind the axis. After the oil has been applied, set the mill in manual mode and use the *X, Y,* and *Z* keys to move the axes back and forth to distribute the oil evenly.

INTRODUCTION TO STUDENT ACTIVITIES

The following section contains a brief description of each activity, the tests and review questions, test and review question answers and blank student answer sheets.

Skills Acquired

Upon completion of each activity the students will have acquired the skills listed below.

Activity 1:

The students will be able to:

- start *Mill Level 4* software.
- open and close part program files.
- use the Tool Bar quick buttons.
- change a part program from G and M codes to conversational codes.
- compile a part program.
- emulate a part program.
- change the view of an emulation.
- step through an emulation.

Activity 2:

The students will be able to:

- open the Program Setup window.
- properly load stock in the mill.
- start the 5600 CNC Mill.
- select a tool movement rate or increment.
- position the end mill at the PRZ
- set the PRZ.
- download a CNC part program.
- mill stock with a CNC part program.

Activity 3:

The students will be able to:

- enter conversational machine codes in the editor window.
- save a part program.
- compile the part program.
- emulate the part program.
- use an alternative method to set the PRZ.
- mill a part from the CNC program.

Activity 4:

The students will be able to:

- edit a part program.
- delete commands.
- copy and paste commands.
- program diagonal lines.
- program a change in cut depth.
- correct errors in a part program.

Activity 5:

The students will be able to:

- program circles.
- program arcs.
- change from move mode to arc mode.
- specify start coordinates.
- specify center coordinates.
- specify end coordinates.

Activity 6:

The students will be able to:

- circular pocket.
- partial pockets.
- cups.
- drill holes.

Activity 7:

The students will be able to:

- program a tic-tac-toe game board in the absolute mode.
- program an X in the incremental mode.
- program an O in the incremental mode.
- copy and paste the code for the X on different program lines.
- copy and paste the code for the O on different program lines.

Activity 8:

The students will be able to:

- call subroutines in a part program.
- edit a part program containing subroutines.
- use the absolute and incremental modes with subroutines.
- replace the end mill.

Activity 9:

The students will be able to:

- design their own machine parts.
- include lines, circles, arc, pockets, and cups in your designs.
- write part programs for the designs using the absolute and incremental modes, milling cycles, and subroutines.
- mill the machine parts.

Equipment and Materials Needed for Student Activities

All of the activities require the use of the:

- 5600 CNC Mill Level 4 Student Guide
- Level 4 Mill software
- Lab-Volt Automation 5600 CNC Mill

All of the activities except Activity 1 require the following materials.

- The tool kit containing all the tools required is provided with each mill.
- A 1/8-in. end mill.
- 2" x 2" x 0.5" machinable acrylic block
- 1.875" x 1.875" x 0.75" support block (shim of wood or other material to raise and support the block to be milled).

Getting the Students Started

Start the course by introducing the CNC mill and describing some of its history. (A short history of the CNC Mill is given in the beginning of the student guide.) The pretest is given before the students start the activity. Each activity requires approximately one class period to be taught (including the reviews where applicable). Reviews are included after Activities 1, 3, 5, and 8. A posttest is included to assess the students' understanding and retention of the material, and is given upon completion of all activities.

ACTIVITY 1: PRE-MACHINING SKILLS

This activity shows the students how to access the Level 4 Mill software. The students will acquire the skills to use the software with a part program.

These skills are the foundation for writing and testing a part program with *Mill Level 4* software. This is the first step to understand CNC mill programming and operation.

Objective

When the student completes this activity, they will have the knowledge and skills to use *Mill Level 4* software.

The student will be able to:

- start *Mill Level 4* software.
- open and close part program files.
- use the Tool Bar quick buttons.
- change a part program from G and M codes to conversational codes.
- compile a part program.
- emulate a part program.
- change the view of an emulation.
- step through an emulation.

Key Points

- The student starts the *Mill Level 4* software from the Windows 95, 98, or the NT 4.0 Desktops by clicking the **Start** button, then selecting **Programs**, then **Lab-Volt Apps**, and lastly clicking on **Mill Level 4**. Or if there is **Mill Level 4** icon on the Desktop, they can double click on it to open the *Mill Level 4* software.

- To open a File, launch the *Mill Level 4* software then click on **File** on the Menu Bar, then **Open**. In the File, Open window, the student can select the part program file to be opened. Or click the **Open** button on the Tool Bar of the Mill Level 4 screen.

- When a mill part program is open in the editor window, the student can toggle between the G and M codes and their conversational code equivalents in the Edit pull-down menu or by pressing the *F2* on the keyboard.

- Compiling a program tests the program for errors and verifies it for emulation or machining. All part programs must be compiled before they can be downloaded to the mill or emulated.

- All part programs should be emulated from several views before actual production. This ensures that the machining will occur without errors or tool crash.

- There are have six view buttons (front, top, side, 3D, zoom out, and zoom in) available under View on the Mill 3D Emulator window.

- The View Control window contains buttons to change the angle and rotation of the view. It also has a **Cut View Toggle** button to permit the student to see only the blue end mill feed rate lines.

- *Mill Level 4* software allows the students to step through a part program emulation line by line so you can easily identify at what point in the program that each tool action occurs.

- The student closes the *Mill Level 4* software by clicking **File** on the Menu Bar and then **Exit**.

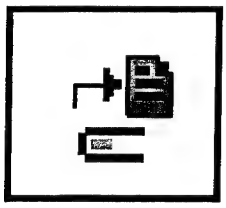
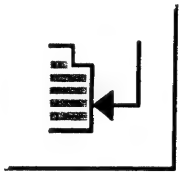
Quiz and Answers: Activity 1 Pre-Machining Skills

NOTE: Correct answers appear entirely in bold print.

1. How do you start the *Mill Level 4* software from the Windows desktop?

   a. Click on the **Start** button, then **Run**, type "Mill Level 4", and then click on **OK**.
   b. Double click **My Computer** icon, and the select the **Mill Level 4** icon.
   c. Click on the Start button, then point to Programs, then Lab-Volt Apps, and click on Mill Level 4.
   d. Click on the **Start** button, point to **Favorites**, and click on **Mill Level 4**.

2. Which of the following buttons on the Mill Level 4 Tool Bar can be used to open the File, Open window?

   a.  b.  c.  d. 

3. How do you switch a part program listing in the editor window from ISO machine codes to conversational codes?

   a. Select **Save As** from the **File** menu and type "conversational".
   b. Compile the program.
   c. Press the *F3* key.
   d. Select Conversational from the Edit menu.

4. What does compile a part program mean?

   a. Run the program in virtual mode.
   b. Stack the program to be run between two others.
   c. Test the program for errors and prepare it for emulation or downloading to the mill.
   d. Add one program to another.

5. What does the Mill 3D Emulator do?

   a. It is used to draw a design on a mill stock using the mouse.
   b. The emulator compiles a part program.
   c. It performs a graphic simulation of the part program.
   d. The emulator downloads a compiled part program to the mill.

6. What does the Mill 3D Emulator window display?

   a. The G code being emulated.
   b. The spindle speed.
   c. The estimated time to mill the stock.
   d. All of the above.

7. What occurs when you click the **Trace** button  on the Mill 3D Emulator window?

   a. The simulation is displayed in dotted lines for easy viewing.
   b. Only lines with negative Z values are displayed.
   c. The emulation is displayed one command of the part program per mouse click.
   d. The last emulation is traced over.

8. The view control for the Mill 3 D Emulator is accessed by

   a. clicking View Control from the Mill menu.
   b. double clicking on the word **View**.
   c. right clicking while the cursor is on the green *Start* button.
   d. pressing F2 on the keyboard while in the Mill 3D Emulator screen.

9. The View Control window can be used to

   a. rotate the emulation view and change the camera viewing angle.
   b. zoom in and out of the emulation view.
   c. start the emulation
   d. All of the above.

10. To exit the Mill 3D Emulator window, press the

   a.  button in the Control box
   b.  button in the Control box.
   c.  box in the upper right corner.
   d. None of the above.

ACTIVITY 2: MACHINING A PART

This activity will show the student how to open a CNC part program a file and automatically machine that part on the Lab-Volt 5600 CNC Mill. This is relevant because in most industrial applications the machinist uses programs stored on disk.

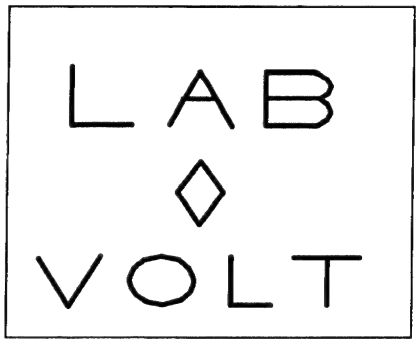
Programmers create a library of part programs so that a specific program can be retrieved to mill a part. The program is written once and then is available for mass production of the part.

Objective

The students completes this activity, they will have the knowledge and skills to machine a part with the Lab-Volt Automation 5600 CNC Mill.

They will be able to:

- open the Program Setup window.
- properly load stock in the mill.
- start the 5600 CNC Mill.
- select a tool movement rate or increment.
- position the end mill at the PRZ
- set the PRZ.
- download a CNC part program.
- mill stock with a CNC part program.



Part to be Milled

Key Points

- Before the part can be milled, the student must know the stock type and size. The student enters the stock type and size as well as other necessary information about the part program in the Program Setup window of the *Mill Level 4* software.
- Any time the stock is less then 1 inch thick, the student should use a block of wood to support the stock. This keeps the stock up above the surface of the vise and prevents the stock from being pushed down into the vise during machining.
- A part program must be compiled before it can be emulated or downloaded to the mill.
- The student should emulate all part programs from several views before they are milled to be sure the design fits on the stock and that there are no tool crashes.

- Before downloading the part program for machining, the programmed reference zero (PRZ) needs to be set on the mill stock.
- The manual mode offers incremental distances as choices for the student to move the end mill to accurately set the PRZ.
- When the end mill is positioned at the correct PRZ, a value of 0 needs to be assigned to each axis. This can either be done one axis at a time, or all three can be assigned a zero value simultaneously.
- Before downloading a part program, the student should check the communications settings in the Remote Setup window of the *Mill Level 4* software.
- On the computer, the student presses the Download button  to download the part program. The same command is also available under Mill on the Menu Bar.
- The Serial Download window of the *Mill Level 4* software and the mill control panel display indicate the status of the download.
- When the part program download is complete, the student presses the *ENTER* key on the mill control panel to machine the part.
- When milling is complete, the mill returns to the Home Position. The default Home Position is 1 inch above the PRZ.
- After the student has turned off the mill power, they'll remove the milled part, then thoroughly clean the 5600 CNC Mill cabinet of any stock scraps or filings before leaving the workstation area.

Quiz and Answers: Activity 2 Machining A Part

1. To load a one half inch thick piece of stock in the 5600 CNC Mill, you should

   a. lay it in the vise and tighten it with a hex wrench.
   b. place it on a support block that is clamped tightly in the vise.
   c. lay it on a support block in the vise and tighten the vise with a hex wrench.
   d. Remove the vice with a hex wrench and clamp the stock to the mill table with the T-bolt.

2. The Programmed Reference Zero (PRZ) is

   a. the point on the block from which all absolute movements are measured.
   b. the point on the block from which all incremental movements are measured.
   c. a starting point defined in the part program.
   d. always at the bottom right of the block.

3. The PRZ is set by

   a. pressing *ZERO, ZERO* on the mill control panel, after you position the end mill is in the PRZ position.
   b. entering "0" values for all three axes under Backlash in the Remote Setup window.
   c. entering the stock size and material in the Program Setup window.
   d. pressing *ZERO* on the mill control panel, and then pressing the 0 key two times.

4. The order of steps to mill stock is to

   a. emulate, compile, set the PRZ, download, and mill the part.
   b. set the PRZ, download, compile, emulate, and mill the part .
   c. download, set the PRZ, emulate, compile, and mill the part.
   d. **compile, emulate, set the PRZ, download, and mill the part.**

5. To download a part program to the mill

   a. click the Download button on Mill Level 4 screen, then select Download from the Main Menu on the mill control panel.
   b. select Download from the Main Menu on the mill control panel, and. then click on the Emulate button on the Mill Level 4 screen.
   c. click the Download button on the Mill Level 4 screen, click OK. on the Serial Download window, and then select Remote from the Main Menu on the mill control panel.
   d. select Remote from the Main Menu on the mill control panel, click the Download button on the Mill Level 4 screen, and click OK on the Serial Download window.

ACTIVITY 3: WRITING A PART PROGRAM - STRAIGHT LINES

In this activity the student will learn how to write a new part program. The part will have a border 0.25 inches around the edge.

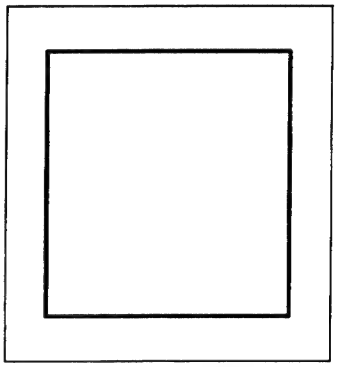
The student should concentrate on the steps necessary to write the part program. Learning how to program straight lines is where all student CNC programmers must start.

Objective

When the student completes this activity, they will have the knowledge and skills to write a part program that uses only straight lines in the design.

They will be able to:

- enter conversational machine codes in the editor window.
- save a part program
- compile the part program.
- emulate the part program.
- use an alternative method to set the PRZ.
- mill a part from your CNC program.



Part to be Milled

Key Points

- The student starts a new part program by clicking on File on the Menu and then New. The Program Setup window appears and the student enters the data for the program and part.

- After the student completes the Program Setup window, a blank editor screen appears with the name of the program in the title bar.

- A backslash at the start of the program line allows the student to enter comments into your program. Comments present information about the part or program, or identify the different sections of a program.

- After the backslash comment lines, the student enters code lines for end mill feed rate (FEED 5), spindle speed (SPEED 1500), and a code to move the end mill in a straight line (MOVE).

- The student uses absolute coordinates to write the border program. That means that all points in the program are measured from the PRZ, X0 Y0 Z0.

- The student refers to a design grid that shows the border design on the entire piece of stock. In order to program the border, the student needs to know the coordinates (X, Y and Z values) of the four corners. These coordinates are used to program the end mill to move in the proper X, Y, and Z directions to cut the border design.

- The code on the last line of the program, END, is used to end the program and send the end mill to the home position.

- The student must save the program before it is complied.

- The student compiles the border program, or any part program, to check for errors.

- The student loads a piece of acrylic (2 in. x 2 in. x 5 in.) over a wooden support block into the mill.

- In some cases, zeroing-out the axes at the PRZ may be difficult. Two examples of these cases are when the view of the corner of the stock is obstructed, or when the stock is irregular in shape. Use an alternative method of setting the PRZ in these cases.

- The student downloads the border part program to the mill and machines the part.

- The student cleans the mill cabinet of any stock scraps or filings before leaving the workstation area.

Quiz and Answers: Activity 3: Writing a Part Program - Straight Lines

1. How do you open a new part program file from the Mill Level 4 Menu?

   a. Click File and then Open.
   b. Click the  button.
   c. Click on the Start button, point to Favorites, click on Mill Level 4 and then New.
   d. Click *File* and then *New*.

2. The X, Y, and Z stock size values in the Program Setup window

   a. must be 0.2 in. larger than the stock to be milled to account for the vice backlash.
   b. must be equal to the size of the stock.
   c. must be 0.1 in. smaller than the stock to be milled.
   d. must be greater than the thickness (Z value) of the stock by 0.5 inches.

3. What is the purpose of the backslash (\) at the beginning of a program line?

   a. It directs the end mill to move diagonally.
   b. It contains comments; not machine code.
   c. It directs the end mill to stop.
   d. It indicates the end of the program.

4. Which of the following lines of code is written correctly?

   a. X1 Y0.5 Z-0.11658
   b. X 1 Y 0.5 Z -0.1
   c. X1 Y0.5 Z-0.1
   d. Z -0.1 X1 Y0.5

5. When you use the alternative method to zero-out the Y-axis for the PRZ, you place the end mill

   a. on top of the stock and enter a Y value equal to the diameter of the end mill.
   b. next to the front of the stock, and enter a negative Y value equal to half the diameter of the end mill.
   c. next to the side of the stock, and enter a minus Y value equal to half the diameter of the end mill.
   d. at the X0, Y0, Z0 position on the stock, and press the *Zero* key on the mill control panel twice.

ACTIVITY 4: EDITING A PART PROGRAM – DIAGONAL LINES

In this activity the student will learn how to edit a part program. Two diagonal lines are added to the program created in Activity 3. The lines produce an X inside the border, and one of the lines has a varying depth.

The student will be shown how to produce of diagonal lines by changing the coordinate values of two axes. Also, they will change the values of three axes to create a diagonal line with varying depth.

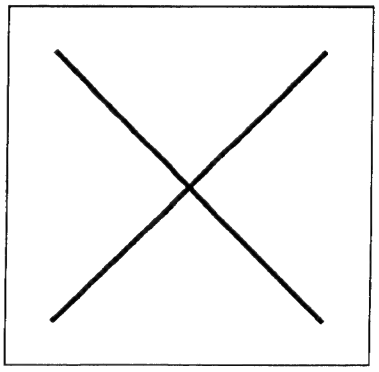
Programming diagonal lines and varying the depth of a cut by changing the coordinate values of two or more of the axes is the next step in learning CNC programming.

Objective

When the student has completed this activity, they will have the knowledge and skills to edit a part program that they create in Activity 3 by adding two diagonal lines to produce an X inside the border.

They will be able to:

- edit a part program.
- delete commands.
- copy and paste commands.
- program diagonal lines.
- program a change in cut depth.
- correct errors in a part program.



Part to be Milled

Key Points

- The student edits an existing part program, and makes changes or additions to the program.
- The student adds commands to the BORDERXX part program that mills two diagonal lines between the corners of the border.
- To delete a command in the editor window, the student will highlight the command and then press the *Delete* key on the keyboard.
- To copy and paste an existing command to a new program line, the student will highlight the command, click the Copy button, place the cursor at the new line, and click the Paste button.

- A command line such as GO1 X.25 Y.25 will move the end mill in a diagonal direction if the start points are X1.75 and Y1.75. This occurs because both the X and Y coordinates are changed from their start points.
- A command such as X1.75 Y.25 Z-.1 will move the end mill in a diagonal direction and change the depth of cut because a minus z value was used.
- The student will learn how to correct errors that maybe found when a program is compiled.
- The student will learn to save a program before it is compiled.
- A side-view emulation will show the student how the depth of a cut varies.
- The student will position the end mill at the PRZ, then press the *Zero* key (on the mill control panel) twice to set all three axes to zero.
- The mill control panel displays Download Complete after a part program is downloaded successfully.
- The student will be instructed to wear safety glasses whenever running the mill.
- The student will learn to clean the work area after they have completed their lesson.

Quiz and Answers: Activity 4: Editing a Part Program - Diagonal Lines

1. How do you delete a line in the editor window of the *Mill Level 4* software?

   a. Place the cursor on the command and then click the **Cut** button on the Menu.
   b. Place the cursor after the command, and keep pressing the *Delete* key until you delete the command.
   c. Highlight the command and then press the *Delete* key on the keyboard.
   d. Place the cursor before the command and keep pressing the *Backspace* key until you delete the command.

2. How do you copy a command in the editor window and paste it at a new line?

   a. Highlight the command, click the **Paste** button, place the cursor at the new line, and click the **Copy** button.
   b. Click the **Copy** button, highlight the command, place the cursor at the new line, and click the **Paste** button.
   c. Place the cursor on the command, place the cursor at the new line, and click the **Paste** button.
   d. Highlight the command, click the Copy button, place the cursor at the new line, and click the Paste button.

3. If you are programming with absolute coordinates in inches, what will the mill do in response to the command X1.5 Y0.3?

   a. move in a straight line to the grid coordinates X equals 1.5 in. with a cut depth of 0.3 in.
   b. move in a diagonal line to the grid coordinates X equals 1.5 in. and Y equals 0.3 in.
   c. move in a diagonal line 1.5 in. in the X direction and 0.3 in. in the Y direction from the last position of the end mill.
   d. move in a diagonal line to the grid coordinates X equals 1.5 in. and Y equals 0.3 in. and end the part program.

4. What command line will move the end mill from the PRZ in a diagonal line and change the cut depth?

   a. GO1 X1.75 Y.25 Z-.1
   b. X1.75 Y.25 DEPTH Z-.1
   c. X1.75 Y.25 RAPID
   d. X1.75 Y.25 GO1 Z-.1

5. Before compiling a part program,

   a. emulate the part program in the top view.
   b. download the part program to the mill.
   c. save the part program to your hard drive.
   d. correct any errors and recompile.

ACTIVITY 5: PROGRAMMING CIRCLES AND ARCS

The students will learn how to create a part program that contains circles and arcs. An arc is a segment of a circle.

The student will also use the conversational programming commands ARC.CW (clockwise arc), ARC.CCW (counterclockwise arc), and ARC (circle) to mill an eye design. To program circles and arcs they must know the coordinates for the starting, center and end points of an arc or circle.

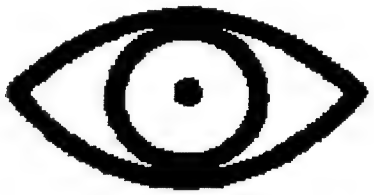
Learning to program arcs and circles is crucial for the students to learn in order to program all the shapes necessary for the design of a complex part.

Objective

Upon completion of this activity, the students will have the knowledge and skills to write a part program that contains circles and arcs.

You will be able to:

- program circles.
- program arcs.
- change from move mode to arc mode.
- specify start coordinates.
- specify center coordinates.
- specify end coordinates.



Part to be Milled

Key Points

- To program the eye, which is composed of a circle and two arcs, the student needs to know the value of six points, the start, center, and end points.
- The ARC.CW (G02) command puts the end mill in the arc mode and moves the end mill clockwise.
- The ARC.CCW (G03) command puts the end mill in the arc mode and moves the end mill counterclockwise.
- ARC (G05) is the command for a circle. It specifies that the start and end points of the arc are the same.
- For an arc, the student must specify the end point.

Quiz and Answers: Activity 5: Programming Circles and Arcs

1. What is the first step in programming an arc or circle?

   a. Use the ARC command.
   b. Move the end mill to the start point for the circle or arc.
   c. Specify the center of the circle or arc.
   d. Use the MOVE command.

2. What is the second step in programming an arc or circle?

   a. Use the END command.
   b. Put the program in the arc mode with the command ARC.CW or ARC.CCW.
   c. Specify the center of the circle or arc.
   d. Use the ARC command for a circle or the end point for an arc.

3. If the start point for a second arc is different than the first arc's end point, what command must you use to get to change the movement mode of the mill?

   a. ARC.CCW
   b. RAPID
   c. ARC
   d. MOVE

4. What is the third step in programming an arc or circle?

   a. **Specify the center of the circle or arc.**
   b. Use the MOVE command.
   c. Use the ARC command.
   d. Use the RAPID command.

5. What is the final step in programming a circle?

   a. Use the RAPID command.
   b. Specify the center of the circle.
   c. Use the ARC command to specify the end point.
   d. Use the MOVE command.

ACTIVITY 6: MACHINING CYCLES

This activity teaches the student how to use canned machining cycles. A canned cycle is a sequence of pre-programmed commands or routines initiated by a single G code. Canned cycles allow you to create pockets, cups, and multiple drill holes without programming each individual end mill movement.

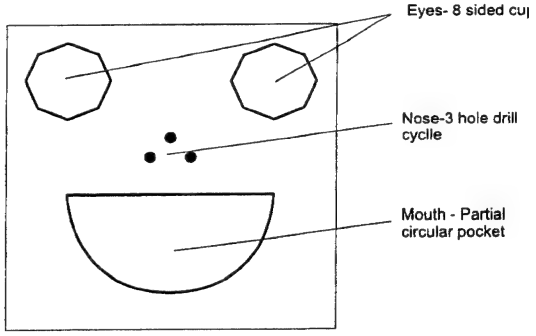
The student will start these canned machining cycles by using the Pocket (G83), Cup (G84), Drill (G81) and Peck (G82) commands. Canned cycles are powerful tools for a programmer and save valuable development and manufacturing time.

Objective

Upon completion this activity, the student will have the knowledge and skills to write a part program that contains canned machining cycles. They will also learn how to use the Mill Level 4 help windows and menus to find information on CNC commands.

The student will be able to program canned cycles for:

- circular pocket.
- partial pockets.
- cups.
- drill holes.



Face Design to be Milled

Key Points

- The *Mill Level 4* software incorporates the *Windows 95, 98,* or *NT 4.0* Help utility that provides information on programming.
- The Help Topics: Mill Level window contains three tabs: Contents, Index, and Find. The Contents dialog shows a menu of three books that contain help information.
- The Glossary windows provide detailed information about each G- code and M-code cycle. The windows contain parameter information, descriptions of the cycles, and both graphic and text examples.
- The student can access information about G- and M-code commands directly from the *Mill Level 4* software Editor window by placing the cursor after the command line and pressing the *F1* key on the keyboard.

- The student uses canned machining cycles to save programming time. A canned cycle is a sequence of preprogrammed commands or routines initiated by a single G-code.
- Canned cycles allow the student to create cuts such as pockets, cups, and multiple drill hole without programming each individual end-mill movement.
- Parameters are properties or values that determine how a command is executed.
- Both the POCKET and CUP commands require two parameters on the same line. The student specifies the number of sides and depth of each end-mill pass. The program uses this information to calculate the number of end-mill passes needed to reach the final pocket or cup depth.
- Pockets and cups can be full or partial and are programmed as if they were arcs. They require start, center, and end points.
- Using the Drilling cycle saves programming time by eliminating the need to program every line required to raise and lower the end mill when drilling holes.

Quiz and Answers: Activity 6: Machining Cycles

1. What is a canned machining cycle?

   a. a sequence of preprogrammed commands for the information in the Program Setup window
   b. a sequence of preprogrammed commands initiated by a single G-code
   c. a programming cycle built into the 5600 CNC *Mill Level 4* software
   d. a programming routine built into the 5600 CNC Level 4 Mill to download a part program

2. What is the quickest way to access the information on a CNC command from the Editor window in the *Mill Level 4* software?

   a. Click on the **Help** button ?.
   b. Look up the command in the G- and M-codes section of the Student Manual.
   c. Refer to the Lab-Volt *Mill Level 4 Software* section of the Student Manual.
   d. Place the cursor on the program line that contains the command and press the *F1* key on the computer keyboard.

3. What two parameters are required for the POCKET programming line?

   a. number of sides and pocket depth
   b. start point and pocket radius
   c. pocket depth and radius
   d. number of sides and pocket diameter

4. Which of the following CUP programming lines specifies the center of the cup?

   a. X.375 Y1.375 Z.03 RAPID
   b. CUP 8 .05
   c. X.375 Y1.625
   d. ARC

5. What type of a shape does the end mill cut when the program lines specifying a Cup Milling Cycle end with the ARC command?

   a. a cylindrical cut in the stock
   b. a half-spherical cut in the stock
   c. a cut that drills 3 holes
   d. a cut that resembles a mouth in a face design

ACTIVITY 7: ABSOLUTE AND INCREMENTAL PROGRAMMING

In this activity, the student will learn how to program using the incremental mode to specify end tool movements. In previous activities, the students used the absolute mode.

In the absolute mode, the student specifies all end tool movements from the PRZ. In the incremental mode, the student specifies end tool movements from its existing location. The advantage of the incremental mode is that program lines can be reused to cut the same shape in other locations on the stock.

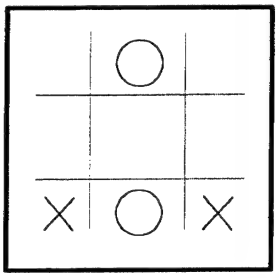
The student writes a part program that will mill a tic-tac-toe game on the stock. They use the absolute mode for the horizontal and vertical lines that form the tic-tac-toe game board. To program the, or moves, they use the incremental mode. By programming the X's and O's in the incremental mode, they can simply copy and paste the code to different locations in the part program to produce additional X's and O's at different locations on the tic-tac-toe game board.

Objective

Upon completion of this activity, the student will have the knowledge and skills to write a part program using both the absolute and the incremental modes.

They will be able to:

- program a tic-tac-toe game board in the absolute mode.
- program an X in the incremental mode.
- program an O in the incremental mode.
- copy and paste the code for the X on different program lines.
- copy and paste the code for the O on different program lines.



Design to be Programmed

NOTE: This activity has three procedures. The last two are optional.

Key Points

- In the absolute programming mode, measure all points from the PRZ.
- In the incremental programming mode, measure all points from the previous point.
- To mill this part, the student uses the absolute mode for the horizontal and vertical lines to form the tic-tac-toe game board.
- To program the Xs and Os, moves, the student uses the incremental mode.
- When programming the moves in the incremental mode, the student copies and pastes the code to different locations in the part program.
- The advantage of using the incremental mode is the ability to easily duplicate a set of program lines in order to cut the same shape in multiple locations.
- The part program is saved before it is compiled o emulated.
- The INCR command placed at the end of a line tells the end mill to make the move in relationship to its current position.
- The student uses the **Copy** and **Paste** functions to copy program lines and paste them in new locations in the Editor window.
- The **Cut View Toggle** button in the View Control window enables the Cut View mode, which simplifies the view of a complex emulation.

Quiz and Answers: Activity 7: Absolute and Incremental Programs

1. What is the purpose of the incremental mode in part programming?

   a. You can specify distances in the part program from the PRZ.
   b. It permits you to automatically save the part program after every incremental program line.
   c. It permits you to enter one command or increment at a time.
   d. You can specify distances in the part program from the current location of the end mill.

2. If absolute is set as the default mode in the Program Setup window, how do you change to the incremental mode for a program line?

   a. You must change to the incremental mode in the Program Setup window.
   b. You must begin the program line with INCR.
   c. You must end the program line with INCR.
   d. You must use G- and M-codes--not conversational codes--in the program line.

3. What is an advantage of using the incremental mode to program a specific shape or design in a part program?

   a. Incremental program lines can reduce programming time.
   b. You can use the incremental program lines again for the shape or design at another location in the part program.
   c. Copying incremental program lines ensures that the shape or design is always the same.
   d. All of the above are correct.

4. If the end mill is at coordinates X1.0, Y0.75, and Z.15 where will it move in response to the program line X.5 Y.25 INCR?

   a. X.5, Y.25, and Z.15
   b. X1.25, Y1.25, and Z.15
   c. X 1.5, Y 1.0, and Z.15
   d. X 1.5, Y 1.0, and Z.3

5. On complex designs, the emulation is clearer if

   a. you use the cut view toggle selection in the view control window.
   b. all the lines with negative x, y, and z values are displayed in the emulation.
   c. you use the **Copy** and **Past** buttons to show the program one command at a time.
   d. you use the INC command.

ACTIVITY 8: SUBROUTINES

This activity teaches the student how to use subroutines in a part program. A subroutine is a program within a program. It is a series of commands that the student call up and execute as part of a larger program.

If a milling design contains several copies of the same cut design or shape, a subroutine for that shape needs to be written. Each time the shape is used, call the subroutine instead of programming the same shape over and over. Using subroutines is similar to the way that the student produced the additional X's and O's in Activity 7. Instead of copying and pasting the program lines for the shape, the student calls a subroutine that includes the program lines for the shape.

Most subroutines are written using the incremental mode so they are not dependent on the PRZ.

An experienced programmer will build a library of subroutines to produce commonly used cut designs or shapes. In industry, this saves valuable time by allowing the programmer to combine multiple subroutines to produce a complex design.

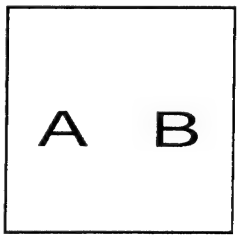
In the procedure, the student will edit the Alpha.m4 program to write a part program to cut your initials into the stock. The Alpha.m4 program contains a subroutine for each letter of the alphabet.

Objective

Upon completion of this activity, the student will have the knowledge and skills to use subroutines in a part program.

They will be able to:

- call subroutines in a part program.
- edit a part program containing subroutines.
- use the absolute and incremental modes with subroutines.
- change the end mill.



Milled Part

NOTE: This activity contains four procedures. The last one is optional.

Key Points

- A subroutine is a label (value) given to a series of commands that can be viewed as a program within a program. Subroutines are called to execute procedures that are part of a larger program.

- If a design requires the end mill to make the same shape several times, a subroutine is written.

- The command (statement) SUB indicates the beginning of a subroutine and associates the subroutine with a specific label. In this way, programmers can initiate the subroutine again by using the label with a CALL statement.

- Subroutines end with the statement END.SUB.

- To allow the use of subroutines in different part programs, they are written in the incremental mode so they are not dependent on the PRZ.

- The student specifies the execution of a subroutine with the conversational CALL or the M97 statement. First the statement appears, then a blank space (or spaces) followed by the label of the subroutine appears.

- An experienced programmer builds a library of subroutines to produce commonly used shapes or end-mill movements.

- Build a stored resource of useful machine code by saving successful subroutines.

Quiz and Answers: Activity 8: Subroutines

1. What is a subroutine?

   a. a program that you execute before another program
   b. a sequence of commands executed as part of a larger program
   c. a program that you request with the **Start** button
   d. a mini part program for small milled parts

2. What statement do subroutines begin with?

   a. BEGIN SUB
   b. SUB BEGINS HERE
   c. START SUB
   d. SUB

3. What statement do subroutines end with?

   a. EXIT.SUB
   b. END SUB
   c. STOP SUB
   d. END.SUB

4. Subroutines are usually written in

   a. incremental mode.
   b. absolute mode.
   c. subroutine mode.
   d. call mode.

5. What statement(s) specifies the execution of a subroutine?

   a. START SUB
   b. CALL
   c. EXECUTE SUB
   d. All of the above are correct.

ACTIVITY 9: ON YOUR OWN

This activity gives the students the opportunity to design their own machine parts, and write part programs. Before the student start this activity, they should have successfully completed Activities 1 through 8.

Because each student programmer's part design will be different, the student will not receive detailed instructions in the procedure. The steps in the procedure are a general guide to help the student design, program, and mill the part.

The student starts off with a simple design and advance to more complicated designs. The first design could consist of vertical and horizontal straight lines, diagonal lines, and possibly a circle using the absolute coordinate mode.

The next design could consist of circles and arcs with some straight lines using both the absolute and incremental modes. The next design could incorporate pockets, cups, and subroutines.

After the student programs a portion of the design, emulate that portion to determine if the emulation matches the design. Frequent emulations of small portions of the design will eliminate mistakes that may be hard to find after the complete design is programmed.

Objective

Upon completion this activity, the student will have the knowledge and skills to mill parts that they designed from part programs they wrote.

They will be able to:

- design your own machine parts.
- include lines, circles, arc, pockets, and cups in your designs.
- write part programs for your designs that use the absolute and incremental modes, milling cycles, and subroutines.
- mill the your machine parts.

Key Points

- The student starts off with a simple design and then advances to more complicated designs.
- The student draws milling design on a design grid showing all coordinate values of the cuts and shapes.
- On the first few lines of the students part program, they will enter program comment lines, and on the following lines, set the feed rate and spindle speed.
- Then they enter the MOVE command and coordinate values to move the end mill to the start of the first cut.
- On the next few lines, they enter commands to cut each individual cut and shape in the design.
- They raise the cutter (+Z) when moving from one shape element to the next and to lower the cutter (-Z) when cutting the shape.
- As the student programs each portion of the design, emulate that portion to determine if the program matches the design. Frequent emulations eliminate mistakes.
- On the last line of the program, the END command is entered, this sends the end mill to the home position.
- The student emulates the final program in several views before milling. This confirms that the design fits on the stock and that the end mill is not going too deep at any point in the procedure.

Quiz and Answers: Activity 9: On Your Own

1. When designing a machine part for milling, the first thing you do is

   a. complete the Program Setup window.
   b. draw a design grid for your machine part.
   c. open the *Mill Level 4* software program.
   d. install the correct end-mill size for milling your part.

2. What should the first several lines of your part program contain?

   a. feed rate and spindle speed data
   b. absolute or incremental mode commands
   c. the PRZ coordinates
   d. program comment lines

3. What follows the first MOVE command in your part program?

   a. a command to place the end mill at the start point of the first cut
   b. the FEED command to set the end-mill cut speed
   c. the SPEED command to set the spindle speed
   d. the ABSOLUTE command to set all end-mill references to the PRZ

4. When writing a part program, you should frequently

   a. recheck your previous program lines.
   b. save and emulate the part program.
   c. determine if you are in the absolute or incremental mode.
   d. compile the part program.

5. You should only use the RAPID command when

   a. cutting a straight line.
   b. the end mill is less than 0.1 in. into the stock.
   c. the cutting speed is set for less than 2 ipm.
   d. the end mill is out of the stock.

APPENDIX A TABLE OF CONTENTS

APPENDIX A: TESTS, ANSWER SHEETS, AND ANSWER KEYS

5600 CNC Mill Pre- and Post Test

1. CNC stands for

   a. Canned Number Cycle.
   b. Computer Numerical Control.
   c. Controlled Numerical Computer.
   d. Card Numerical Control.

2. The large red button on the mill control panel is the

   a. *Start* button.
   b. *Stall Light Override* button.
   c. *Emergency Stop* button.
   d. *Resume Operation* button.

3. The PRZ is set at

   a. The left front corner of the workpiece, just touching the surface.
   b. The back right corner of the workpiece, just touching the surface.
   c. The front right corner of the workpiece, just touching the surface.
   d. The center point of the workpiece, just touching the surface.

4. When machining on the mill you must always

   a. download a program.
   b. use an end mill.
   c. wear your safety glasses.
   d. run in absolute mode.

5. The 5600 CNC Mill has three axes:

   a. A, B, and C.
   b. X, Y, and Z.
   c. X, Y, and 0.
   d. G, M, and Z.

6. When you use the *Level Four Mill Software* to run in "absolute mode," this means

   a. that the cutter moves first along the X-axis and then along the Y-axis.
   b. that each point in the program is measured from the last point.
   c. that all points in the program are measured from the PRZ (0,0,0).
   d. that all points in the program are measured from the center of the block.

7. Incremental mode is used for programming

   a. program subroutines and repeated shapes.
   b. the PRZ and the dimensions of the workpiece.
   c. the spindle speed and feed rate.
   d. CAD/CAM software.

8. To Compile a program means

   a. add one program to another.
   b. test the program for errors and prepare it for machining.
   c. to run the program in virtual mode.
   d. stack the program to be run between two others.

9. The Programmed Reference Zero (PRZ) is

   a. The point on the block from which all absolute movements are measured.
   b. The point on the block from which all incremental movements are measured.
   c. A starting point defined in the part program.
   d. Always at the bottom right of the block.

10. The MOVE command does not have to be reentered each time a new move coordinate is entered.

    a. True.
    b. False.

11. Cutting lines always have a negative Z value if programmed in absolute mode.

    a. True.
    b. False.

12. To machine a line: move to the start point of the line, lower the cutter, enter the MOVE command, and then the (X,Y) value of the line endpoint.

    a. True.
    b. False.

13. A canned cycle is

    a. a machining cycle that is halted.
    b. a sequence of commands initiated by a single G-code.
    c. manually inputted coordinates.
    d. a part program that is down loaded via a satellite link.

14. To machine a circle: move to the start point of the circle, lower the cutter, enter the ARC command and the center point of the circle.

    a. True.
    b. False.

15. The POCKET command requires two parameters on the same line as the command.

    a. True.
    b. False.

16. The CUP command requires three parameters on the same line as the command: the number of sides, center point of the cup, and depth of the cup.

    a. True.
    b. False.

17. To create a subroutine to be used in various programs, enter the SUB command.

    a. True.
    b. False.

18. To signal the end of a program, enter the FINISH command.

    a. True.
    b. False.

19. PRZ stands for

    a. Present Reference Zone
    b. Programmable Relief Zoning
    c. Programmable Reference Zero
    d. Positional Reference Zero

20. Always turn the mill off before cleaning or servicing.

    a. True.
    b. False.

5600 CNC Mill Pre- and Post Test Answer Sheet

NAME: _____ **DATE:** _____

CLASS: _____

Write the answers in the space provided. Hand in the answer sheet when complete.

1. ___
2. ___
3. ___
4. ___
5. ___
6. ___
7. ___
8. ___
9. ___
10. ___
11. ___
12. ___
13. ___
14. ___
15. ___
16. ___
17. ___
18. ___
19. ___
20. ___

5600 CNC Mill Pretest Answer Key

1. b
2. c
3. a
4. c
5. b
6. c
7. a
8. c
9. a
10. a
11. a
12. a
13. b
14. b
15. a
16. b
17. a
18. b
19. c
20. a

5600 CNC Mill Pre- and Post Test with Answers

1. CNC stands for

   a. Canned Number Cycle.
   b. Computer Numerical Control.
   c. Controlled Numerical Computer.
   d. Card Numerical Control.

2. The large red button on the mill control panel is the

   a. *Start* button.
   b. *Stall Light Override* button.
   c. *Emergency Stop* button.
   d. *Resume Operation* button.

3. The PRZ is set at

   a. The left front corner of the workpiece, just touching the surface.
   b. The back right corner of the workpiece, just touching the surface.
   c. The front right corner of the workpiece, just touching the surface.
   d. The center point of the workpiece, just touching the surface.

4. When machining on the mill you must always

   a. download a program.
   b. use an end mill.
   c. wear your safety glasses.
   d. run in absolute mode.

5. The 5600 CNC Mill has three axes:

   a. A, B, and C.
   b. X, Y, and Z.
   c. X, Y, and 0.
   d. G, M, and Z.

6. When you use the *Level Four Mill Software* to run in "absolute mode," this means

   a. that the cutter moves first along the X-axis and then along the Y-axis.
   b. that each point in the program is measured from the last point.
   c. that all points in the program are measured from the PRZ (0,0,0).
   d. that all points in the program are measured from the center of the block.

7. Incremental mode is used for programming

   a. program subroutines and repeated shapes.
   b. the PRZ and the dimensions of the workpiece.
   c. the spindle speed and feed rate.
   d. CAD/CAM software.

8. To Compile a program means

   a. add one program to another.
   b. test the program for errors and prepare it for machining.
   c. to run the program in virtual mode.
   d. stack the program to be run between two others.

9. The Programmed Reference Zero (PRZ) is

   a. **The point on the block from which all absolute movements are measured.**
   b. The point on the block from which all incremental movements are measured.
   c. A starting point defined in the part program.
   d. Always at the bottom right of the block.

10. The MOVE command does not have to be reentered each time a new move coordinate is entered.

    a. True.
    b. False.

11. Cutting lines always have a negative Z value if programmed in absolute mode.

    a. True.
    b. False.

12. To machine a line: move to the start point of the line, lower the cutter, enter the MOVE command, and then the (X,Y) value of the line endpoint.

    a. True.
    b. False.

13. A canned cycle is

    a. a machining cycle that is halted.
    b. a sequence of commands initiated by a single G-code.
    c. manually inputted coordinates.
    d. a part program that is down loaded via a satellite link.

14. To machine a circle: move to the start point of the circle, lower the cutter, enter the ARC command and the center point of the circle.

    a. True.
    b. False.

15. The POCKET command requires two parameters on the same line as the command.

    a. True.
    b. False.

16. The CUP command requires three parameters on the same line as the command: the number of sides, center point of the cup, and depth of the cup.

    a. True.
    b. False.

17. To create a subroutine to be used in various programs, enter the SUB command.

    a. True.
    b. False.

18. To signal the end of a program, enter the FINISH command.

    a. True.
    b. False.

19. PRZ stands for

    a. Present Reference Zone.
    b. Programmable Relief Zoning.
    c. Programmable Reference Zero.
    d. Positional Reference Zero.

20. Always turn the mill off before cleaning or servicing.

    a. True.
    b. False.

Safety Test

The following Safety Test appears in the 5600 CNC Mill Student Manual.

1. The large red button located on the mill control panel is the manual speed override.

   a. True
   b. False

2. One way to halt the execution of a part program is to press the *Stall Light Override* button.

   a. True
   b. False

3. Always wear safety goggles while operating the mill.

   a. True
   b. False

4. The safety door can remain open while operating the mill if the operator wears safety goggles.

   a. True
   b. False

5. Never leave the key in the *Emergency Stop* button.

   a. True
   b. False

6. Never leave the mill unattended while it is running.

   a. True
   b. False

7. Always turn the mill off before cleaning or servicing.

   a. True
   b. False

8. If pushed, the *Emergency Stop* button can be released simply by pulling it.

   a. True
   b. False

9. Never leave any tools inside the mill cabinet.

   a. True
   b. False

10. Know what to do in an emergency situation before operating the mill.

    a. True
    b. False

Safety Test and Answers

The following Safety Test appears in the 5600 CNC Mill Student Manual. In this Instructor Guide, the correct answers are shown in bold print.

1. The large red button located on the mill control panel is the manual speed override.

   a. True
   b. False

2. One way to halt the execution of a part program is to press the *Stall Light Override* button.

   a. True
   b. False

3. Always wear safety goggles while operating the mill.

   a. True
   b. False

4. The safety door can remain open while operating the mill if the operator wears safety goggles.

   a. True
   b. False

5. Never leave the key in the *Emergency Stop* button.

   a. True
   b. False

6. Never leave the mill unattended while it is running.

   a. True
   b. False

7. Always turn the mill off before cleaning or servicing.

   a. True
   b. False

8. If pushed, the *Emergency Stop* button can be released simply by pulling it.

   a. True
   b. False

9. Never leave any tools inside the mill cabinet.

   a. True
   b. False

10. Know what to do in an emergency situation before operating the mill.

    a. True
    b. False

5600 CNC Safety Test Answer Sheet

NAME:_____ **DATE:**_____

CLASS:_____

Write the answers in the space provided. Hand in the answer sheet when complete.

1. ___

2. ___

3. ___

4. ___

5. ___

6. ___

7. ___

8. ___

9. ___

10. ___

5600 CNC Safety Test Answer Key

1. B
2. A
3. A
4. B
5. A
6. A
7. A
8. B
9. A
10. A

Activity Quiz Answer Sheet (Activity One)

ACTIVITY NUMBER:_____ **DATE:** _____

NAME: _____ **CLASS:** _____

Write the answers in the space provided. Hand in the answer sheet when complete.

1. ___

2. ___

3. ___

4. ___

5. ___

6. ___

7. ___

8. ___

9. ___

10. ___

Activity Quiz Answer Sheet

ACTIVITY NUMBER:_____ **DATE:** _____

NAME: _____ **CLASS:** _____

Write the answers in the space provided. Then hand in the answer sheet.

1. ___
2. ___
3. ___
4. ___
5. ___

Activity 1 Pre-Machining Skills Quiz Answer Key

1. C
2. A
3. D
4. C
5. C
6. D
7. C
8. A
9. A
10. C

Activity 2 Machining a Part Quiz Answer Key

1. B
2. A
3. A
4. D
5. D

Activity 3 Writing a Part Program - Straight Lines Quiz Answer Key

1. B
2. B
3. B
4. C
5. B

Activity 4 Editing a Part Program - Diagonal Lines Quiz Answer Key

1. C
2. D
3. B
4. A
5. C

Activity 5 Programming Circles and Arcs Quiz Answer Key

1. B
2. B
3. D
4. A
5. C

Activity 6 Machining Cycles Quiz Answer Key

1. B
2. D
3. A
4. C
5. A

Activity 7 Absolute and Incremental Programming Quiz Answer Key

1. D
2. C
3. D
4. C
5. A

Activity 8 Subroutines Quiz Answer Key

1. B
2. D
3. D
4. A
5. B

Activity 9 On Your Own Quiz Answer Key

1. B
2. D
3. A
4. B
5. D

APPENDIX B: FREQUENTLY ASKED QUESTIONS

1. How do you free the axis when it's at its limit?

- From the Manual screen, press *ZERO, ESC*. Return to the Main Menu. Select Manual. Use the appropriate *X, Y,* or *Z* key to move away from the limit.

- If the axis is still jammed, press *ENTER* twice from the Manual menu. This puts the system in incremental mode; i.e., pressing the axis key moves it the displayed distance. Refer to **Distance** in **Appendix B: Manual Mode of Operation**. The axis limit switches are disabled in incremental mode so that pressing the appropriate *X, Y,* or *Z* key will free the axis.

> **CAUTION:** Be especially careful when in incremental mode! Because the limit switches are disabled, extra care is required to avoid damaging the mill by going past its limits.

- Press *ESC* to reset the switches.

2. How do you reset the *EMERGENCY STOP* button?

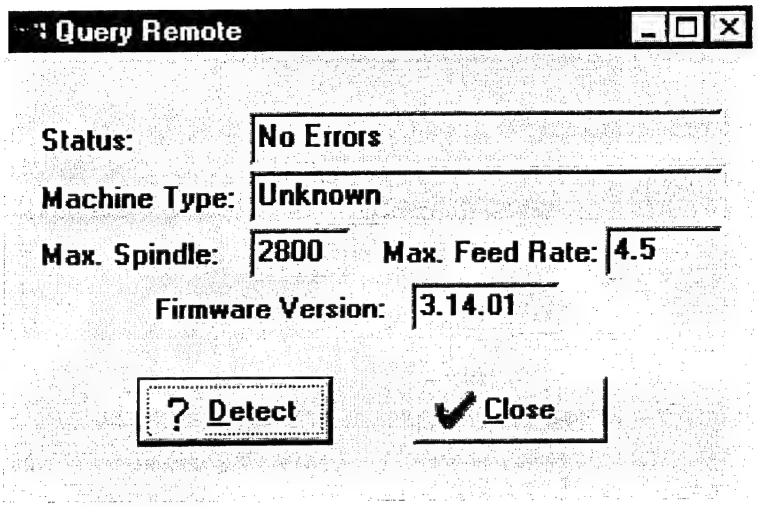
   Insert the *EMERGENCY STOP* key in the lock and turn it.

3. How do you choose feed rates and spindle speeds?

   The choice of feed rates and spindle speeds depends on the material to be milled and the intricacy or fineness of the pattern to be milled. In general, the harder the surface, the slower the cut needs to be made and, therefore, the slower the feed rate. The spindle speed needs to be set fast enough to not leave rough edges, yet slow enough so that the heat generated by the cutting doesn't melt the material being milled.

4. How do you debug communications?

   Select Remote from the Main Menu on the mill control panel. On the PC, select the **Mill** menu, then **Query Remote**.



Query Remote

Press **Detect** to attempt a connection.

If the connection is correct, the machine type, maximum spindle speed, feed rate and firmware version number are confirmed or updated.

If errors are discovered, here are some suggestions of things to check:

- Verify that the serial cable is connected properly to both the mill's 9-pin SERIAL INTERFACE port and the computer's 9 or 25-pin communication port.
- Verify that the computer's communications port being used for cable connection to the mill is correctly designated in the Level 4 Remote Setup screen – for example, if the computer port is labeled COM1, make sure COM1 is selected on the Remote Setup.
- Verify that the computer serial port is working. Try testing the mouse on that port.
- In Windows, ensure that the selected communications port is configured as 9600, 8, N, 1.
- In some cases, old firmware versions can cause communications failure. Ensure that the mill firmware version is 3.02 or higher.

APPENDIX C: MANUAL MODE OF OPERATION

In manual mode, all operations of the 5600 CNC Mill can be controlled using the keys on the mill control panel. Manual mode of operation is only available from the Manual menu.

```
Man R 14.000 ipm
X   0.000  Sp  0  RPM
Y   0.000  I0000 S0000
Z   0.000  O0000
```

Manual Menu

The first line of the Manual menu is a status line. The "Man" indicates manual mode. The next alphanumeric sequence indicates the feed rate of the mill axis. An "R" indicates that the number following it is the rapid speed. In the example above, the rapid feed rate is 14 inches per minute. This means that the axis will travel at the rapid speed unless another feed rate is entered on the Feed Rate menu (described below under Feed Rate).

The numbers following the X, Y, and Z indicate the coordinates of the current position of the three axes (the X, Y table and the headstock).

The "Sp 0 RPM" indicates the current spindle speed.

The lower right of the screen is where the status of the input/output from each of the four TTLs and four solenoids is displayed. These input/outputs are set in the Output menu and the Accessories menu. The four numbers following each letter can be ones (1) or zeros (0). The position of the number within the series indicates which port is displayed; i.e., the third number indicates the status of port 3. Zero indicates that the status of the port is low, or not being used. The number "1" indicates that the status is high, or set to be used. The zeros following the I indicate that the TTL inputs are low; those following the O indicate that the TTL outputs are low; and those following the S indicate that the solenoid outputs are low.

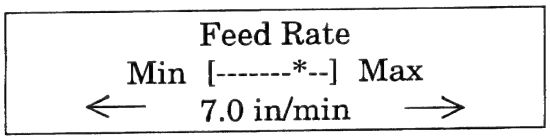
For simplicity when describing feed rates and distances, the mill table and/or the headstock is referred to as the axis. To move the axis, press the key for the direction in which it is to be moved. For example, to move the mill table to the right, press *-X*. To move the headstock up, press *+Z*. The table or headstock will move as long as the key is depressed. Optionally, the distance the table or headstock is to move can be set so that each time an axis direction key (e.g., *-X*) is pressed the axis moves that distance. Refer to Distance described below.

Zero the Mill

To zero the mill is to inform the mill of the cutter starting point from which all movements are to be referenced. Use the *X, Y,* and *Z* keys to position the cutter at the desired starting point and press the *Zero* key twice to set the zero there. More detailed information is given in the body of this manual under ***Set the Programmed Reference Zero.*** If, during operations, adjustments in the zero need to be made, access the Zero Menu by pressing the *Zero* key followed by the axis key of the direction to be zeroed. Pressing the negative axis key (*-X, -Y, -Z*) indicates the number to be entered is a negative number. Pressing the positive axis key (*+X, +Y, +Z*) indicates the number to be entered is a positive number. The number can be entered by pressing the number keys on the control panel and then pressing *ENTER*. The number entered is in the same units as the feed rate. If the wrong number is entered accidentally, simply return to the Manual Menu and go through the zero procedure again to enter the correct number.

Feed Rate

The feed rate for manual mode can be set from the Feed Rate menu which can be accessed by pressing the *Feed Rate* key from any menu.



Feed Rate Menu

The Feed Rate menu is a bar menu allowing the feed rate to be set in 1 inch/minute increments by using the *-X* and *+X* keys. Press *ENTER* to set the feed rate or press *ESC* to exit the Feed Rate menu without making any changes.

In manual mode, the easiest way to change from the set feed rate to the rapid rate is by pressing *ESC* to return to the Main Menu then *ENTER* to return to the Manual menu which automatically resets the feed rate to R (rapid).

Distance

An axis can be moved the distance determined by the length of time that a direction key is held down or a specified distance can be selected from the Manual menu.

> **CAUTION:** Operating in incremental mode (moving a distance specified in the Manual menu) disables the limit switches. Be careful not to damage the mill by going past its limits.

A selection of distances at the designated feed rate is accessible on the first line of the Manual menu by pressing the *ENTER* key. The first time the *ENTER* key is pressed, the feed rate set in the above Feed Rate procedure is displayed. Each additional time the *ENTER* key is pressed the distance is displayed and increased by preset increments.

```
0.100 F   7.000 ipm
X  0.000  Sp 0 RPM
Y  0.000  I0000 S0000
Z  0.000  00000
```

Distance Display in Manual Menu

The first number in the line is the distance that the axis will move when the axis direction key is pressed. In the example above, the distance is 0.100 inches. The units of the distance are the same as the units displayed for the feed rate. The second number in the line is the feed rate, the rate at which the axis will move; e.g., 7 inches per minute. The mill only moves the distance displayed for each key press. For example, if the *-X* key is pressed while the distance is set at 0.1 inches, the mill table will only move to the right 0.1 inches. To move the table an additional 0.1 inches, the key must be released and pressed again.

Standard (inches)	0.001	0.005	0.010	0.050	0.100	0.500	1.000
Metric (milli-meters)	0.01	0.05	0.1	0.5	1.00	5.00	10.00

Selectable Distances

Spindle Speed

The spindle speed of the mill can be set from the Manual, Setup, or Output menus by pressing the *Spindle Speed* key.

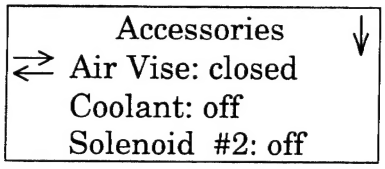
```
Spindle Speed
Off [*---------] Max
<-      0 RPM      ->
```

Spindle Speed Menu

The Spindle Speed menu is a bar menu allowing the spindle speed to be set in 100 rpm increments by using the *-X* and *+X* keys. The spindle speed increases (or decreases if *-X* is pressed) each time the *+X* key is pressed while in this menu. If the *Zero* key is pressed, the speed drops to zero. Press *ENTER* to set the spindle speed or press *ESC* to exit the Spindle Speed menu without making any changes.

Accessories

The accessories connected to the mill can also be controlled from the mill control panel. The Accessories menu can be accessed by pressing the Main Menu key from the Manual menu. It allows the user to turn the accessories on or off without leaving the Manual menu. There are four options. Each is a toggle choice selected by pressing *-X* or *+X*.



Accessories Menu

The first selection, Air Vise, can be either "closed" (the default status) or "opened". The other three selections, Coolant, Solenoid #2, and Solenoid #3 can be turned "off" (default) or "on".